Dear Editor,

We are writing to express our gratitude for the insightful and constructive comments provided by the reviewers for our manuscript titled *"IMEX_SfloW2D v2: a depth-averaged numerical flow model for volcanic gas-particle flows over complex topographies and water"* (Preprint gmd-2023-80). We are pleased to inform you that we have carefully considered the feedback and suggestions provided by Reviewer 1 and Reviewer 2 and have made substantial revisions to our paper in accordance with their recommendations.

Reviewer 1's comments were particularly insightful, highlighting the need for a more comprehensive discussion of some of the present limits of the model (vertical profiles, settling velocity). In response, we have expanded the Discussion section to provide a more in-depth explanation of these limitations and possible ways to overcome them in future versions. Additionally, Reviewer 1 recommended that the role of sedimentation compared to deposition be better distinguished. To address this, we have changed the terminology used in the manuscript and revised the text to clarify this point.

Reviewer 2's feedback was invaluable in identifying areas where our data analysis and interpretation could be strengthened. Specifically, Reviewer 2 pointed out that for the benchmark 1D tests presented in the work, there are no analytical solutions to prove that the model reproduces them correctly or to help the reader visualize the model's accuracy. As per this suggestion, we have included the equations for the analytical solutions in the manuscript and added the relevant plots to the figures. These changes help to appreciate the accuracy of the numerical model we implemented.

In addition to these major revisions, we have meticulously addressed numerous minor comments and suggestions from both reviewers, ranging from improvements in clarity and coherence to the correction of typographical errors.

We would also like to inform you that one of the authors, Samantha Engwell, is affiliated with the British Geological Survey (BGS). As per the policies of the BGS, all scientific papers authored by BGS employees undergo an internal review process to ensure alignment with institutional guidelines and objectives. For this manuscript, it a landslide expert was engaged from the BGS who suggested additional changes that were made.

We believe that the revisions made in response to the reviewers' feedback have substantially enhanced the quality and impact of our manuscript. We are confident that the changes we have implemented align well with the standards of rigor and clarity that GMD upholds.

Enclosed herewith, please find the detailed answers to all reviewer's comments, the list of new references added to the manuscript, the source of the Matlab script used to compute the analytical solution of the 1D test cases, and a revised version of our manuscript highlighting all the suggested changes. We kindly request that you consider our revised submission for publication in GMD. We would also like to extend our gratitude to the reviewers for their time, expertise, and dedication in helping us improve the quality of our work.

Thank you for considering our revised manuscript. We eagerly await your decision and remain at your disposal for any further information or clarification.

Sincerely,
Mattia de' Michieli Vitturi
Tomaso Esposti Ongaro
Samantha Engwell

Answers to RC1

Q. *Line 20: "...yes, reliable"*

A. The text has been modified as suggested.

Q. *Lines 44-45. Agreed. However, some words should be spent on how a shallow-water model, which by definition cannot solve for vertical gradients of the flow properties, can be employed to simulate DPDCs, which are strongly vertically-stratified (particle concentrantion, flow density, flow velocity).*

A. The following text has been added:

> "In particular, for currents where turbulent mixing is large enough to maintain vertically uniform concentration, shallow water models provide a good approximation (Bonnecaze et al., 1993). This approach can still be used for vertically stratified flows, but it is necessary to introduce appropriate correction factors into the equations, generally based on a simplifying assumption of well-developed and stationary vertical profiles and yet not easy to calculate explicitly (Biagioli et al., 2021; Keim and de' Michieli Vitturi, 2022)."

Q. *Lines 66-74. I agree that entrainment is a key factor controlling DPDCs, however the role of sedimentation should be equally emphasized in this introduction since, like entrainment, it controls the rate of change of the flow bulk density and, hence, its existence. Indeed, some words should be spent in emphasizing the concept that a DPDC, as any other density current (e.g., turbidity currents), exists until there is a density contrast with the ambient in which it flows* **A.** The following text has been modified as follows:

> "On par with entrainment, sedimentation exerts a major role in controlling the rate of change of the flow bulk density and, hence, its existence. Indeed, dilute PDCs, as any other density current (e.g., turbidity currents), exist until there is a positive density contrast with the ambient in which they flow. If sufficient air is entrained or a sufficient amount of particles are lost by sedimentation, then the density falls below that of the ambient air and buoyant liftoff occurs.
>
> The capability of dilute flows to overcome topographic barriers, and consequently the sedimentation regimes, also depends on the Richardson number (Woods and Wohletz, 1991; Woods et al., 1998). For these reasons, an accurate description of these regimes is mandatory for a model of dilute PDCs, both in the definition of the equations and in their numerical discretization and solution."

Q. *Lines 91-92. The repository lacks of a User manual (or at least I cannot find it), which would be very beneficial.*

A. For all the examples provided with the code in the Github repository there is a README file with detailed instruction to create the input files, to run the simulation and to post-process the results.

Q. *Line 104. Is this fixed in the code? Can another gas specie be used? If yes, I would generalize here*

A. The gas component are not fixed, and we tried to make it more clear in the text, by modifying the line indicated by the Reviewer in the following way:

> "we assume that the flow is a homogeneous mixture of a multi-component gas phase (for the applications presented in this work air and water vapour)"

We also added the following text in Section 3.1

"The specific gas constants are provided in the code as user inputs, and more gas can be added, allowing for the simulation of a mixture of any number of gas components."

Q. *Lins 151-152. What are the consequences of this approximation and why is it reasonable to take this into account?*

A. Some of the consequences were already stated before Eq.6 but, to make it more clear, we revised the text and added some references which can provide a more detailed analysis, which is out of the scope of this paper. We report here the revised text (original text from line 138 to 156):

"It is worth noting that the design of conservative and stable numerical schemes for the solution of Eqs. (2-5) requires some care. This is because the numerical solution of mass and momentum equations, even when these quantities are globally conserved, does not necessarily result in an accurate description of the mechanical energy balance of the shallow water system (Fjordholm et al., 2011; Murillo and García-Navarro, 2013). In fact, many numerical schemes perform well in practice but they may have an excessive amount of numerical dissipation near shocks, preventing a correct energy dissipation property across discontinuities (which can arise even in the case of smooth topography). A quantification of the error in the conservation of mechanical energy is beyond the scope of this paper, also because the error is case dependent and the topography plays a crucial role, but the reader can refer to Arakawa and Lamb (1977); Arakawa (1997); Arakawa and Lamb (1981); Fjordholm et al. (2011) for a comprehensive discussion of this issue. In general, to guarantee energy conservation in smooth regimes, it is desirable to design high-order schemes adding a minimal amount of numerical dissipation. Numerical errors in the computation of the mechanical energy can lead to errors associated with the mixture temperature obtained from the total mixture specific energy and the kinetic energy computed from mass and momentum equations. For this reason, in some cases, instead of the full energy equation as presented above, it is preferable to solve a simpler transport equation for the specific thermal energy $C_v T$:

$$\frac{\partial}{\partial t}\left(\rho_m h C_v T\right) + \frac{\partial}{\partial x}\left(C_v T \rho_m h u\right) + \frac{\partial}{\partial y}\left(C_v T \rho_m h v\right) =$$

$$-\sum_{i_s=1}^{n_s}\left(\rho_{s,i_s} C_{s,i_s} T S_{s,i_s}\right) + C_a \rho_a T_a E_a + C_{wv} \rho_{wv,b} T_b E_{wv},$$

We remark that in this equation we neglect heating associated with friction forces. This term can be important for some applications where viscous forces are particularly large, for example lava flows, but are negligible for the applications presented in this work. It is also worth noting that the numerical solution of Eqs. (2-4,6) does not guarantee that the total energy is conserved. In the following section, for simplicity, we use the temperature equation to demonstrate how the system of equations can be written in a more compact form and in the description of the numerical schemes, but for the applications presented in this work the full energy equation is solved."

Q. *Lines 167-168. So, the assumption is made that, everything that is sedimenting is also creating a deposit. This is not always necessarily the case, since sedimenting particles (in fact, I would use a sedimentation term in the conservation equations, instead of a deposition term) can be moved as a traction carpet or even re-entrained.*

A. We thank the reviewer for pointing out that sedimentation does not always result in a deposit. We included the observation of the reviewer in the text, and we also clarified that Eq. 14 is not used in the applications presented in this work, but can be used for different applications. The following text has been added after Eq. 14:

> "This equation assumes that sedimentation of particles immediately results in an increase in deposit thickness, which is not always the case, because particles in the basal layer that forms at the bottom of the flow could be moved as a traction carpet or even re-entrained. While Eq. (14) is appropriate for flows that can be modeled by the system of conservation equations (10), as for example for lahars, it is not used in the application to dilute PDCs presented in this work."

Q. *Lines 169-173. I would further emphasize the strength of IMEX, i.e. the fact that one can choose among 6 different rheological models, hence 6 different ranges of applications, with the same code thanks to the way the model is developed.*

A. The following sentence has been added to the text:

> "This rheological model is available in IMEX_Sflow2D v2, together with a plastic rheology (Kelfoun, 2011), a temperature dependent friction model (Costa and Macedonio, 2005) and a lahar rheology (O'Brien et al., 1993), largely increasing the range of applicability of this version of the code."

Q. *Lines 177-179. As far as I understand reading the manuscript, IMEX is not restricted to radially spreading flows. In my opinion this should be stated more clearly.*

A. We tried to make it more clear modifying the text in the following way:

> "The code we present is mostly aimed at simulating 1D/2D flows, for which we do not simulate the initial phase and, most importantly, for which in radially spreading flows the front velocity decreases because of the increasing radius."

Q. *Line 211. This equation is from Parker et al. (1987), I see the paper is not cited here. https://doi.org/10.1080/00221688709499292 Furthermore, there are other relationships available in the literature, e.g. https://doi.org/10.1029/2003JF000052, https://doi.org/10.1029/2019GL084776. While these should not necessarily be implemented into the code, they can be at least cited perhaps in the introduction when you first introduce the entainment. But at least the model you use should be cited.*

A. We followed the suggestion of the reviewer by adding the suggested references and modifying the text in the following way:

> "where ϵ, following Bursik and Woods (1996), is the entrainment coefficient given by Parker et al. (1987):
>
> $$\epsilon = \frac{0.075}{(1 + 718Ri^{2.4})^{0.5}}. \quad (1)$$
>
> We remark that, even if not implemented in the model, there are other relationships available in the literature for the entrainment coefficient (Ancey, 2004; Dellino et al., 2019a)."

Q. *Lines 226-228. Any reference for this assumption? The larger fraction for smaller particles is reasonable and can perhaps be explained by the larger specific surface (hence, also the type of particles (e.g., pumice vs. lithic fragments) via the shape)*

A. We added the reference Dufek et al. (2007), which provide this estimation.

Q. *Lines 234-237. Sedimentation does not necessarily coincide with deposition. I suggest the authors to review the use of terminology. The models described below and the sink term in the conservation equations are sedimentation rate terms, deposition rates may differ because there may be processes at the deposition interface that mobilize or even re-entrain the particles settling onto the ground. Furthermore, the use of a model like eq. 19 has the limitation that it does not take into account the effect of the flow, specifically the turbulence, on the sedimentation rate. Models like eq. 19 assumes a constant sedimentation rates that does not take into account the turbulence, which can keep particle in suspension. This depends on the Rouse number. See for example the recent works of Dellino et al. (https://doi.org/10.1111/sed.12485, https://doi.org/10.1111/sed.12693) and cited literature therein. I am not implying that the authors should implement these models (which may add a further complexity) but suggesting to make these limitating assumptions clear in the text.*

A. We thanks the reviewer for the comment, and we added the following text at the end of the section:

> "The sedimentation model described by Eq. 19 represents the loss of particles from the flow, but does not necessarily correspond to a deposition rate, i.e. the rate of accretion of deposit thickness. This is true only when the ratio between the actual deposition and the sedimentation rate is high ($> 5 \times 10^{-3}$ according to Shimizu et al., 2019). In fact, there may be processes at the deposition interface that mobilize or even re-entrain the particles settlinginto the concentrated basal layer and then onto the ground. Furthermore, in order not to add further complexity, the adopted equation assumes a constant sedimentation rate which does not take into account the turbulence, which can keep the particles in suspension. A more accurate description of sedimentation based on the Rouse number, representing a ratio of particle settling velocity to scale of turbulence, can be found in (Dellino et al., 2019b, 2020; Valentine, 1987)."

In addition, we changed the notation used for sedimentation from D_{s,i_s} to S_{s,i_s}.

Q. *Lines 241-252. Also here, there are more recent and accurate treatments of the particle settling velocity calculations. For simple spheres (which is a strong and limiting assumptions, see my comments in the following), there are drag laws that avoid the jump in the Lun and Gidaspow drag law (e.g. https://onlinelibrary.wiley.com/doi/10.1002/cjce.5450490403; https://doi.org/10.1016/0032-5910(89)80008-7), which in some situation may cause problems. Then there is the problem of using a drag law for spheres in the case of natural sediments/volcanic particles, which are (very) far from spheres, with strong implications on the Cd and therefore vs. Even if the authors do not want to implement such models (e.g., https://doi.org/10.1016/j.powtec.2018.12.040, https://doi.org/10.1002/2017JB014926, https://doi.org/10.1016/0032-5910(93)80051-B), they should clearly state that their approach is a simplification and that can lead to overestimations of the terminal velocity, hence sedimentation rates in the flow.*

A. We added a paragraph in the section "Conclusion" where the limitations highlighted by the reviewer, together with other limitations, are discussed. We report here the text discussing the settling velocity model:

> "We also point out that the present version of the code implements a rather simplified model for particle settling velocity, and in the future we plan to adopt more recent and accurate treatments for its calculations. For simple spheres, as assumed here, there are drag laws that avoid the jump in the Lun and Gidaspow drag law (Clift and Gauvin, 1971; Haider and Levenspiel, 1989), which can result in numerical problems when the the velocity is computed with an iterative numerical scheme. In addition, the use of a drag law for spheres in the case of natural sediments/volcanic

particles is an important simplification that can lead to overestimation of the terminal velocity, hence the sedimentation rate in the flow. We remark that the adoption of more complex models (Ganser, 1993; Bagheri and Bonadonna, 2016; Dioguardi et al., 2018) would also require the knowledge of additional parameters characterizing the shape of the particles, which are not always easy to retrieve."

Q. *Lines 366-367. Move this to the beginning of the section.*

A. The text has been moved to the beginning of the paragraph.

Q. *Lines 391-392. I suggest moving fig. c under figure a.*

A. The bottom panels have been inverted. In addition, the analytical solutions have been added to the plot to show the accuracy of the numerical solver, together with a legend.

Q. *Lines 497-498. I understand, however I think a complete comparison with a real case, for which data are available, would add more value to the manuscript and the software. The potential user would be more keen to use it for realistic applications. It would be also interesting to see benchmarking applications like comparisons with large-scale experiments, like the experiments carried out recently in New Zealand (PELEE) or the older ones by the University of Bari group. Maybe the former are more complicated for this code, since they produce channelised DPDCs; the latter, though, should be straightforward to reproduce.*

A. Our aim for this paper was to present development of a numerical model for simulating several different types of flows. To demonstrate the capabilities of the model, we chose examples that enabled us to highlight the range of model applications, for example simulating flows over water and flow interaction with topography. While these simulations were informed by real events, they were intended to give the reader insight into the scope of the model capabilities, rather than benchmark the model. With reference to the terminology proposed by Esposti Ongaro et al. (2020), as well as from CFD modeling standards (e.g., Oberkampf and Trucano, 2002) we here focus on the presentation of the model qualification, verification and presentation of unit tests and synthetic benchmarks. We understand the value gained from directly reproducing real events and experiments, but such large- and full-scale benchmark studies are bodies of work in themselves, and are outwith the scope of the paper. In any case, the presented Krakatau runs show the applicability of the code to the natural scale phenomenon. Further work dedicated to the reproduction of flows from the Krakatau 1883 eruption, through application of the model using new, recently collected, grainsize and thickness data from the event is currently in progress. We have changed the text in the abstract, introduction and discussion to highlight that the model has been applied to Krakatau volcano, with model inputs informed by the 1883 eruption, rather than implying that the simulations directly reproduce the eruption.

Q. *Line 499. These are the units of volumetric flow rate.*

A. We changed the units to kg s^{-1}. By double-checking for the units, we also noticed that the reported mass flow rate was wrong and now we reported the correct value (10^{10} kg s^{-1}) in the manuscript.

Q. *Line 500. Space missing.*

A. We added the space.

Q. *Lines 589-590. I expected to see an implementation of the method presented in Biagioli et al. (2021) (https://doi.org/10.1016/j.apm.2020.12.036) in this version of IMEX that allows simulating DPDCs. Is there a reason why this method, that to me looks like a very promising one to take into account vertical profiles in shallow water models, is not taken into account in v2 of IMEX?*

A. The method implemented in Biagioli et al. 2021 focuses on the profiles for Newtonian laminar flow, thus with a parabolic velocity profile, which is not adequate for the turbulent flows considered in this work. In addition, in Biagioli et al. 2021 the flow does not consider the presence of particles. In any case, we agree with the reviewer that the approach presented in

this work is very promising also for other kind of flows, and a similar method has been proposed in Keim and de' Michieli Vitturi (2022), but it is still the object of research. In any case, to account for the comment of the reviewer, we added the following text in the conclusion section:

> "In future versions of the code we plan to adapt an approach similar to that presented for Newtonian laminar flows in (Biagioli et al., 2021), where the depth-average equations have been modified to account for the vertical variation of velocity and temperature. The proposed modifications were implemented in the first version of IMEX-SfloW2D, and the applicability of such approachs to velocity and particle concentration profiles for dilute pyroclastic density currents has been shown in (Keim and de' Michieli Vitturi, 2022)."

Q. *Finally, since this code should guarantee run times compatible with probabilistic volcanic hazard assessments, it would be useful to have some data on the run times vs. used computational resources of a realistic test case (e.g. the presented Krakatau test case).*

A. We added a paragraph in the conclusion section to discuss this point.

> "In conclusion, the depth-averaged model introduced in this study offers a promising avenue for advancing probabilistic volcanic hazard assessment. By providing a computationally efficient alternative to traditional 3D models, it significantly reduces the computational burden while still capturing, as shown by the Krakatau application, essential aspects of volcanic flows. For instance, for the simulation of the Krakatau case study for 1800 seconds, the code required 2 hours of computational time on a 7th Generation Kaby Lake Intel Core i7 processor, which could be substantially reduced with a parallel run on multiple cores. Moreover, the utilization of a High-Performance Computing (HPC) system further amplifies the potential of the depth-averaged model in probabilistic volcanic hazard assessment, enabling the execution of a large number of simulations within a reasonable timeframe. This makes the model well-suited for practical applications where timely hazard assessment is crucial."

Answers to RC2

Q. *Equation 1-4: the notation must be similar. s,is in the equation, sis in the text.*
A. The text has been fixed.

Q. *Line 118: if possible try to avoid the exponent that can be mistaken for "to the b power"*
A. We changed the notation to avoid any confusion.

Q. *Line 143: it's difficult to know whether this simplification has little or a major impact on the results. Under what conditions could induced errors alter the model's results in relation to the natural phenomenon?*

"This can lead to numerical errors associated with the mixture temperature obtained from the total mixture specific energy and the kinetic energy computed from mass and momentum equations. For this reason, in some cases, instead of the full energy equation as presented above, it is preferable to solve a simpler transport equation for the specific thermal energy CvT"

A. This point is important, but a full description of the problem, and a quantification of the error introduced, are out of the scope of this paper, and it is the subject of an active research in the field of the numerical discretization of shallow water equations. In any case, we agree with the reviewer that some additional information should be provided and we have modified the text of the paragraph next to that mentioned by the reviewer in the following way, also providing references for a better comprehension of the problem:

> It is worth noting that the design of conservative and stable numerical schemes for the solution of Eqs. (2-5) requires some care. This is because the numerical solution of mass and momentum equations, even when these quantities are globally conserved, does not necessarily result in an accurate description of the mechanical energy balance of the shallow water system (Fjordholm et al., 2011; Murillo and García-Navarro, 2013). In fact, many numerical schemes perform well in practice but they may have an excessive amount of numerical dissipation near shocks, preventing a correct energy dissipation property across discontinuities (which can arise even in the case of smooth topography). A quantification of the error in the conservation of mechanical energy is beyond the scope of this paper, also because the error is case dependent and the topography plays a crucial role, but the reader can refer to Arakawa and Lamb (1977); Arakawa (1997); Arakawa and Lamb (1981); Fjordholm et al. (2011) for a comprehensive discussion of this issue. In general, to guarantee energy conservation in smooth regimes, it is desirable to design high-order schemes adding a minimal amount of numerical dissipation.

Q. *175: Froude number*
A. Fixed.

Q. *178: "The code we present is mostly aimed at simulating 2D spreading flows, for which we do not simulate the initial phase". Once published and distributed, this code will most likely be used to simulate complete eruptions, including initial phases. Can it be used? Is it possible to quantify the errors if it is used to reproduce the initial phase? Errors of a few percent can be a problem from a mathematical point of view, but are perfectly acceptable in volcanology, where knowledge of source conditions is often limited.*

A. The observation to which the reviewer's comment refers is specific for dilute PDCs. As stated in the manuscript introduction, these kind of flows can form in relation to several volcanic behaviours (Valentine, 1987; Branney and Kokelaar, 2002; Sulpizio et al., 2014; Dufek et al., 2015), including the collapse of Plinian and Vulcanian columns, and the explosive fragmentation of a lava dome or cryptodome (Sigurdsson et al., 2015). To properly describe these source phases, a more complex 3D model would be required, also able to simulate compressible, turbulent flows and non-equilibrium gas-particle dynamics. Inherent to the foundations of the model described

in this work is the assumption that the flow propagates under gravity at the atmospheric pressure, and that the effects of compressibility are negligible. For these reasons, we think that the radial source implemented in the code provides a good initial condition for the propagation of dilute PDCs when they are already at atmospheric conditions, and in particular for flows generated by column collapse of explosive eruptions. For other kind of flows that the model can simulate, but are not presented in this work (landslides, pyroclastic avalanches, lahars, lava flows), the code provides different kinds of initial conditions, which allows simulation of the complete event. In the code repository, the Etna test case provides an example of a pyroclastic avalanche where the flow is generated by the collapse of a crater.

Q. *Equation 16 / line 125: Something is not clear to me. Is the entrainment calculated only on the edges or in the whole flow? From equation 16, it seems that entrainment also takes place within the flow itself. In that case, the hypothesis of line 125 does not seem right to me because the flow carries momentum: "There are no terms associated with air and water vapour entrainment, because they do not carry any horizontal momentum into the flow."*

A. Both air entrainment and water vapour entrainment take place at the interfaces between the flow and the entrained gases. Thus, air entrainment occurs over the whole top surface of the flow, while water vapor entrainment occurs at the bottom surface, when in contact with the sea surface. Entrainment, as any other exchange with external medium, results in mass, momentum and energy exchanges, because the mass, momentum and energy "possessed" before the entrainment "enter" into the flow. But, for both atmospheric air and water vapour, the velocity before entraiment is null, and thus no momentum is added to the flow with entrainment. With regards to energy, no kinetic energy is added, while the thermal energy of entrained air and water vapour increase the total energy of the flow.

Q. *They way chosen to define the Ridchardson number is close to a Froude number (Ri = 1/Fr^2). I am a bit lost to see the link with the ratio of the stabilizing stratification of the current to destabilizing velocity shear and, consequently, with the entrainment."*

A. The reviewer is correct in the relationship he wrote about the Richardson number and the Froude number. We will try to explain here why the Richardson number also quantify the balance between the stabilizing effects of stratification (variation of density with height) and the destabilizing effects of velocity shear (variation of velocity with height) in a fluid flow, thus providing insight into the stability characteristics of the flow and is particularly relevant in situations involving buoyancy forces, such as in the atmosphere and oceans.

To make it more clear we first observe that the Richardson number can be written also in the following equivalent form:

$$Ri = \frac{g}{\rho}\frac{\Delta\rho}{\Delta z}\left(\frac{\Delta u}{\Delta z}\right)^{-2}$$

where:

- g is the acceleration due to gravity.
- ρ is the density of the fluid.
- $\Delta\rho$ is the change in density across a certain distance Δz in the vertical direction.
- Δu is the change in horizontal velocity across the same distance Δz.

Now, let's break down the relationship between the Richardson number and the stabilizing stratification of the current versus destabilizing velocity shear:

1. *Stabilizing Stratification*: When there is a stable stratification, it means that denser fluid is located below lighter fluid, creating a situation where lighter fluid tends to rise over denser fluid due to buoyancy forces. In other words, the density gradient ($\Delta\rho/\Delta z$) in the denominator of the Richardson number quantifies the stabilizing effect of stratification. If this gradient is large (strong stratification), it indicates that the buoyancy forces are significant, and the flow is less likely to become turbulent or mixed due to the upward tendency of the lighter fluid.

2. *Destabilizing Velocity Shear*: Velocity shear refers to the change in velocity across a distance. If there is a significant change in horizontal velocity ($\Delta u/\Delta z$) across the same distance

Δz, it can lead to velocity shear, which can induce mixing and turbulence in the flow. Velocity shear is often associated with the destabilizing effects that can disrupt the stability imposed by stratification.

The relationship between these two aspects becomes apparent when you examine the Richardson number. A large Richardson number (indicating strong stratification relative to velocity shear) means that the stabilizing effects of buoyancy forces dominate over the destabilizing effects of velocity shear. This leads to a more stable flow with less mixing and turbulence, thus a reduced entrainment.

Conversely, a small Richardson number (indicating weak stratification relative to velocity shear) implies that the destabilizing effects of velocity shear are strong enough to overcome the stabilizing effects of buoyancy forces. This can result in increased mixing, turbulence, and enhanced entrainment.

In the manuscript, we tried to summarize what is written above in the following way:

> "As the flow propagates, air is entrained at a rate which: (i) is proportional to the magnitude of the difference in velocity between the flow and the stationary ambient; and (ii) depends on the ratio of the stabilizing stratification of the current ($N^2 = \frac{g}{\rho}\frac{\Delta\rho}{\Delta z}$, where N is the Brunt-Väisälä frequency) to destabilizing velocity shear ($M^2 = \left(\frac{\Delta u}{\Delta z}\right)^{-2}$, where M is also called the the Prandtl frequency). This ratio is expressed by the Richardson number (Cushman-Roisin and Beckers, 2011) and, following Bursik and Woods (1996), is written in the following equivalent form:
>
> $$Ri = g'h/(u^2 + v^2), \tag{2}$$
>
> where g' is the reduced gravity. Written in this form, this is essentially a ratio between between potential and kinetic energies, with the numerator being the potential energy needed to entrain the overlying buoyant fluid and the denominator being the kinetic energy of the flow which cause this entrainment."

Following the suggestion of Reviewer 1, we also added a reference for Eq. 17 (Parker et al., 1987). This reference can help to better understand the relationship between entrainment and the Richardson number.

Q. *289: Could the authors explain what is an "opportune slope limiter"? On what criteria are they chosen? This choice seems to have strong consequences on edge values and therefore on the fluxes.*

A. We added the following text to the manuscript:

> "Slope limiters are employed to mitigate the occurrence of excessive oscillations and unrealistic behavior that might arise during the solution of partial differential equations through finite volume methods, especially in proximity to shocks and discontinuities. When prioritizing accurate shock representation, the Superbee slope limiter can be the best choice as it maintains sharper discontinuities, but at the cost of a tendency of smooth humps to become steeper and squared with time. If preserving monotonicity and minimizing oscillations in smooth regions are more important, then MinMod could be favored. For a more detailed analysis of slope limiters and finite volume methods, the reader can refer to LeVeque (2002)."

Q. *Figure 1: The authors use two flux values, Q(w, j-1, k) and Q(E, j, k) for example. What is the physical reality of these two flows at the same location? I didn't understand whether there was a procedure to make them compatible.*

Line 311: same question here. I wonder if it's physically correct to consider 2 speeds, one to the left, the other to the right, in the same place? Is this choice linked to the physics of the phenomenon or to numerical stability?

A. Having two different values at the two sides of interfaces in a numerical method, such as finite volume methods or finite difference methods, might seem counterintuitive from a physical standpoint, as it doesn't directly reflect the smooth nature of many real-world physical phenomena. However, this discrepancy is a consequence of the mathematical discretization process used in numerical methods to approximate continuous partial differential equations (PDEs) over discrete domains.

In particular, having two values at the two sides of interfaces in finite volume methods (FVM) is essential for properly describing and capturing discontinuous solutions. Discontinuities are abrupt changes or jumps in the solution variables, such as shock waves, interfaces between different materials, or other sharp transitions in physical phenomena. Finite volume methods are designed to handle these types of phenomena, and the presence of two values at cell interfaces contributes to their accuracy in representing such solutions.

Here's how:

- *Accuracy at Discontinuities*: Discontinuities are inherently present in many physical processes, and they need to be accurately captured by numerical methods. Having two different values at the cell interfaces allows FVM to represent the abrupt changes in solution variables, such as density, pressure, or concentration, which occur at these discontinuities.

- *Numerical Fluxes*: FVM calculates numerical fluxes, which represent the flow of quantities across cell interfaces. These fluxes are based on the differences between the values at the two sides of the interface. By incorporating both values, FVM can construct more accurate approximations of how the solution changes across the discontinuity.

- *Shock Capturing*: One of the main strengths of FVM is its ability to effectively capture and resolve shock waves. Shock waves are characterized by steep gradients and rapid changes in solution variables. The presence of two values at the interfaces allows FVM to accurately compute fluxes and gradients, capturing the sharpness of the shock profile.

- *Limiters*: In the presence of steep gradients, slope limiters (as mentioned earlier) control oscillations and ensure the accuracy of the solution. These limiters often use information from both sides of the interface to determine how to adjust gradients and avoid unphysical oscillations.

- *Convergence and Conservation*: Properly resolving discontinuities is crucial for the convergence of the numerical solution to the correct physical solution. FVM's focus on conservation principles within control volumes helps maintain the integrity of these abrupt changes.

In summary, having two values at the two sides of interfaces in finite volume methods is indeed crucial for describing and capturing discontinuous solutions accurately. It allows FVM to effectively handle sharp gradients, shock waves, and other abrupt changes in solution variables that are characteristic of real-world physical phenomena.

To make this clearer in the manuscript, we summarized the above text in the following way in the paper:

> "Having two different values at the two sides of interfaces in a numerical method might seem counterintuitive from a physical standpoint. However, this jump is a consequence of the mathematical discretization process used in the finite volume method to approximate continuous partial differential equations (PDEs) over discrete domains. In particular, having two values at the two sides of interfaces in finite volume methods is essential for properly describing and capturing discontinuous solutions."

Q. *Lines 325-327: I'm not able to understand what these choices imply for the model's results.*

A. We added the following text to the manuscript:

> "The implicit treatment of the friction terms, when dealing with strongly non-linear rheologies, avoids many problems related to the proper stopping condition of the flow, without the need for introducing arbitrary thresholds. In addition, the automatic derivation of the friction term allows for a simple implementation of any rheological model, including formulations dependent on any model parameter."

Q. *Figure 2: captions d and c are inversed.*

A. The figure has been fixed.

Q. *Figure 2, 3, 4... : if I'm not mistaken, the proposed solutions are only those of the model. Is there no analytical solution to prove that the model reproduces them correctly or to help the reader visualize the model's precision? How do the reader know if the model is working properly?*

A. We added in the manuscript the equations for the analytical solutions of test cases shown in Figures 2 and 3, and the relative profiles have been added to the Figures. We report here the text added to the manuscript.

> "By integrating equation 30 with the additional condition of constant volumetric flux (as expressed by the first of Eqs. 28) we also have, in the case of a regular solution, the following Bernoulli relation:
>
> $$\frac{(hu)^2}{2g'h^2} + h + z = Const, \tag{3}$$
>
> which gives us a relation between the topography elevation and the flow height. Using Eq. 30, in the cases of supercritical or subcritical flows without regime transitions, it is possible to obtain the following implicit equation for the flow depth (Delestre et al., 2013):
>
> $$h(x)^3 + \left(B(x) - \frac{(h_0 u_0)^2}{2g'h_0^2} \right) h(x)^2 + \frac{(h(x)u(z))^2}{2g'} = 0, \tag{4}$$
>
> where h_0 is the flow thickness at the inlet or outlet of the domain."

Similar text has been added after the description of the transcritical test case, with and without shock. Furthermore, a Matlab script to compute and plot the analytical solution is provided in this document after the list of new references.

Q. *Figure 10 (13 and the others): if possible, change the curve lines to adapt them to black and white printing*

A. We changed the line patterns in figure 13 to adapt them to black and white. For Figures 10 and 11 this was not easy to do, because of the large number of lines plotted. If the editor thinks it necessary, we can add each of the individual profiles as separate figures in the Supplementary material.

Q. *Figure 13: why at the source, the entrained air represents 100% of the gases? Can't you distinguish between air initially present and entrained air? The scale does not seem right for the low particle concentration. Perhaps a more specific scale on the right-hand axis would be more appropriate.*

A. As explained above, the source does not the correspond to the vent, but to the radial flow of the already decompressed mixture, resulting from the collapse of the explosive column.

In this initial phase, which we do not simulate, a lot of atmospheric air is entrained, and this is why the particle concentration is low. In any case, we want to point out that at the source of the simulated dilute flow, the entrained air does not represent 100% of the gases, because a small amount is also represented by the volcanic gas (here water vapour). To make it clearer that the initial condition refers to source of the radially spreading flow, and not to vent conditions, we added the following text to the manuscript:

> We point out that we do not simulate here the collapse of the explosive column, but only the dilute flow generated by the entrainment of atmospheric air and the collapse of the column. Thus, the initial conditions of the simulation refers to the source conditions of the radially spreading dilute flow.

Q. *Line 500 and 365: kg m-3 (with a space)*
A. The notation has been fixed.

List of new references

Ancey, C.: Powder snow avalanches: Approximation as non-Boussinesq clouds with a Richardson number–dependent entrainment function, Journal of Geophysical Research: Earth Surface, 109, 2004.

Arakawa, A.: Computational design for long-term numerical integration of the equations of fluid motion: Two-dimensional incompressible flow. Part I, Journal of computational physics, 135, 103–114, 1997.

Arakawa, A. and Lamb, V. R.: Computational design of the basic dynamical processes of the UCLA general circulation model, General circulation models of the atmosphere, 17, 173–265, 1977.

Arakawa, A. and Lamb, V. R.: A potential enstrophy and energy conserving scheme for the shallow water equations, Monthly Weather Review, 109, 18–36, 1981.

Bagheri, G. and Bonadonna, C.: On the drag of freely falling non-spherical particles, Powder Technology, 301, 526–544, 2016.

Bonnecaze, R. T., Huppert, H. E., and Lister, J. R.: Particle-driven gravity currents, Journal of Fluid Mechanics, 250, 339–369, 1993.

Clift, R. and Gauvin, W.: Motion of entrained particles in gas streams, The Canadian Journal of Chemical Engineering, 49, 439–448, 1971.

Cushman-Roisin, B. and Beckers, J.-M.: Introduction to geophysical fluid dynamics: physical and numerical aspects, Academic press, 2011.

Delestre, O.: Simulation du ruissellement d'eau de pluie sur des surfaces agricoles, Ph.D. thesis, Université d'Orléans, 2010.

Dellino, P., Dioguardi, F., Doronzo, D. M., and Mele, D.: The entrainment rate of non-Boussinesq hazardous geophysical gas-particle flows: An experimental model with application to pyroclastic density currents, Geophysical Research Letters, 46, 12 851–12 861, 2019a.

Dellino, P., Dioguardi, F., Doronzo, D. M., and Mele, D.: The rate of sedimentation from turbulent suspension: An experimental model with application to pyroclastic density currents and discussion on the grain-size dependence of flow runout, Sedimentology, 66, 129–145, 2019b.

Dellino, P., Dioguardi, F., Doronzo, D. M., and Mele, D.: A discriminatory diagram of massive versus stratified deposits based on the sedimentation and bedload transportation rates. Experimental investigation and application to pyroclastic density currents, Sedimentology, 67, 2013–2039, 2020.

Dioguardi, F., Mele, D., and Dellino, P.: A new one-equation model of fluid drag for irregularly shaped particles valid over a wide range of Reynolds number, Journal of Geophysical Research: Solid Earth, 123, 144–156, 2018.

Esposti Ongaro, T., Cerminara, M., Charbonnier, S.J. et al. A framework for validation and benchmarking of pyroclastic current models. Bull Volcanol 82, 51 (2020). https://doi.org/10.1007/s00445-020-01388-2

Ganser, G. H.: A rational approach to drag prediction of spherical and nonspherical particles, Powder technology, 77, 143–152, 1993.

Haider, A. and Levenspiel, O.: Drag coefficient and terminal velocity of spherical and nonspherical particles, Powder technology, 58, 63–70, 1989.

Keim, B. and de' Michieli Vitturi, M.: Implementation of Vertical Profiles for Velocity and Particle Concentration Into a 2D, Depth-Averaged Model for Dilute Pyroclastic Density Currents, in: AGU Fall Meeting Abstracts, vol. 2022, pp. V52C–0017, 2022.

Kelfoun, K.: Suitability of simple rheological laws for the numerical simulation of dense pyroclastic flows and long-runout volcanic avalanches, Journal of Geophysical Research: Solid Earth, 116, 2011.

LeVeque, R. J.: Finite volume methods for hyperbolic problems, vol. 31, Cambridge university press, 2002.

Parker, G., Garcia, M., Fukushima, Y., and Yu, W.: Experiments on turbidity currents over an erodible bed, Journal of Hydraulic Research, 25, 123–147, 1987.

Shimizu, H. A., Koyaguchi, T. and Suzuki, Y. J. (2023). Dynamics and deposits of pyroclastic density currents in magmatic and phreatomagmatic eruptions revealed by a two-layer depth-averaged model. Geophysical Research Letters, 50, e2023GL104616.

Matlab script for analytical solutions

```
close all
clear all

L = 25;

nx = 2000;

x = linspace(0,L,nx);

% topography
zM = 0.2;
z = 0*x + (x>8).*(x<12).*(zM-0.05*(x-10).^2);

g = 9.81;

%% subcritical solution
figure;
subplot(2,1,1)
plot(x,z)


q0 = 4.42;
hL = 2;

h_sub = 0*x;

for i=1:nx,

    h_set = roots([1,(z(i)-q0^2/(2*g*hL^2)-hL),0,q0^2/(2*g)]);
    [minValue,closestIndex] = min(abs(h_set-hL));
    h_sub(i) = h_set(closestIndex);

end

hold all
plot(x,z+h_sub);
xlim([0,25])
ylim([-0.10,2.1])

subplot(2,1,2);
plot(x,q0./h_sub)

xlim([0,25])
ylim([2.191,2.6072])


%% supercritical solution

figure;
subplot(2,1,1)

plot(x,z)

q0 = 10;
hL = 1;

h_sup = 0*x;

for i=1:nx,

    h_set = roots([1,(z(i)-q0^2/(2*g*hL^2)-hL),0,q0^2/(2*g)]);
```

```
    [minValue,closestIndex] = min(abs(h_set-hL));
    h_sup(i) = h_set(closestIndex);

end

hold all
plot(x,z+h_sup);
xlim([0,25])
ylim([-0.10,1.3])

subplot(2,1,2);
plot(x,q0./h_sup)
xlim([0,25])
ylim([9.7688,10.01])


%% transcritical without shock

figure;
subplot(2,1,1)

plot(x,z)

q0 = 1.53;
hL = 0.66;

h_trans = 0*x;

hC = (q0/sqrt(g))^(2/3);

hOLD = 1.01;

for i=1:nx,

    h_set = roots([1,(z(i)-q0^2/(2*g*hC^2)-hC-zM),0,q0^2/(2*g)]);
    if x(i) < 10
        [minValue,closestIndex] = min(abs(h_set-hOLD));
    else
        [minValue,closestIndex] = min(abs(h_set-(hOLD-0.1)));
    end
    h_trans(i) = h_set(closestIndex);
    hOLD = h_trans(i);

end

hold all
plot(x,z+h_trans);
xlim([0,25])
ylim([-0.10,1.1])

subplot(2,1,2);
plot(x,q0./h_trans)
xlim([0,25])
ylim([1.39,3.88])


%% transcritical wit shock

figure;
subplot(2,1,1)

plot(x,z)

```

```
q0 = 0.18;
hL = 0.33;

h_trans = 0*x;
h1 = 0*x;
h2 = 0*x;

hC = (q0/sqrt(g))^(2/3);

hOLD = 0.4;

fVal = 0*x;

for i=1:nx,

    x(i)
    h_set = roots([1,(z(i)-q0^2/(2*g*hC^2)-hC-zM),0,q0^2/(2*g)])
    if x(i) < 10
        [minValue,closestIndex] = min(abs(h_set-hOLD));
    else
        [minValue,closestIndex] = min(abs(h_set-(hOLD-0.1)));
        [minValue,closestIndex] = max(h_set);
        closestIndex = 2;
    end
    h1(i) = h_set(closestIndex);
    hOLD = h1(i);

    h_set2 = roots([1,(z(i)-q0^2/(2*g*hL^2)-hL),0,q0^2/(2*g)]);

    [minValue,closestIndex] = min(abs(h_set2-hL));
    h2(i) = h_set2(closestIndex);

    if x(i)<10

        fVal(i) = 1.0;
    else

        fVal(i) = q0^2*(1/h1(i)-1/h2(i))+g/2*(h1(i)^2-h2(i)^2);

    end

end

[minValue,closestIndex] = min(abs(fVal));

x_shock = x(closestIndex);

for i=1:nx,

    if x(i)<=x_shock
        h_trans(i)=h1(i);
    else
        h_trans(i)=h2(i);
    end

end

hold all
plot(x,z+h_trans);
xlim([0,25])
ylim([-0.05,0.5])

subplot(2,1,2);
```

```
plot(x,q0./h_trans)
xlim([0,25])
ylim([0.339,2.43])
```