

This is an interesting study on coupling CFD with MPM to model multi-phase mixtures of soils. However, the authors have been ambitious to integrate too many stuffs into a single framework, which causes confusions in many parts of its implementation. The manuscript needs to be thoroughly revised for further consideration. The authors are advised to refer to the following specific comments for revision.

1. Line 89 and anywhere else: It is not real phase transition and please avoid this term. Note that phase transition is used in constitutive models sometimes.

We acknowledge that the terminology "phase transition" can potentially cause misunderstandings. Thus, we have made the necessary revisions to the manuscript as outlined below:

"This transition, where the sediment transforms into fluid-like debris and then re-establishes a porous medium, poses a challenging task for simulating submarine landslides»

2. Line 92: Please give references to support the statement "submarine landslide can be modeled by CFD". It seems weird to use CFD to model soil movement. Moreover, the authors stated that soil constitutive laws cannot be easily applied in the Eulerian framework, but the particle-based methods can overcome this problem. The underlying logic looks weird, and constitutive laws have nothing to do with numerical methods. The authors are suggested to reword the whole paragraph to make a clear comparison between CFD and particle-based methods. Note that the authors need to state that the CFD mentioned here is referred to the FVM one.

We have rephrased the entire paragraph to enhance clarity, as follows:

Due to this transition, submarine landslides can be modeled using either the Eulerian framework or the Lagrangian framework. The Eulerian framework involves calculation of material response at specific time-space points. For instance, methods within Computational fluid Dynamics, example using Finite Volume Methods (FVM) are employed to simulate submarine landslides \cite{CFD1, CFD2, CFD3, CFD4} by solving governing equations in a full-Eulerian framework. While FVM is capable of handling complex flows, such as turbulent flows, it falls short in accounting for the triggering mechanisms of submarine landslides. This is due to the challenge of incorporating 'constitutive laws' of sediment materials within the Eulerian framework. This is particularly significant because converting material time derivatives into fixed space derivatives involves arduous mathematical tasks, especially for soil materials that rely on nonlinear tensor operations and history-dependent state/internal variables. In contrast, the Lagrangian framework, including various particle-based methods, provides a solution to this problem. In the Lagrangian framework, material "particles" are tracked individually through space, and material properties and internal variables are determined at and follows these particles. These methods have been extensively used to simulate landslides, like Material Point Method (MPM) \cite{Tran2019}, Smooth Particle Hydro Dynamics \cite{Capone2010}, Particle Finite Element Method \cite{Zhang2019}, or Coupled Eulerian Lagrangian Method \cite{Dey2016}. For simplicity, these previous simulations have adopted a total stress analysis, neglecting the pore pressure development which is a key factor triggering slope failures.

3. Line 114: other approaches such as SPH can also handle complex fluid flows. Meanwhile, particle-based methods can be parallelized very well. Please comment why FVM/CFD is a better option.

Your insights are spot-on. The choice between Eulerian methods like FVM/CFD and Lagrangian methods, such as particle-based ones, is contingent on the specific simulation context. In the realm of submarine landslide simulations, where intricate fluid flows with turbulence are involved, Eulerian methods have certain advantages over Lagrangian methods.

In situations where turbulence occurs at a very fine resolution, methods based on an Eulerian approach, like FVM/CFD, offer distinct advantages. This is particularly evident in scenarios involving submarine landslides, where fluid flows exhibit complexity and turbulence. In contrast, utilizing Lagrangian methods with numerous dynamic discrete "particles" to model fine resolution of turbulence within a fluid can prove to be immensely computationally intensive. Managing many particles in intricate flow fields, especially when multiple phases are present (e.g., air overlapping with water overlapping porous media), escalates computational demands.

In such circumstances, a Eulerian approach like FVM/CFD demonstrates its suitability. Despite potential limitations such as time step constraints due to grid size considerations for stability and accuracy, Eulerian methods are well-suited to capture the dynamics of fluid flow fields. While our experience of Smooth Particle Hydrodynamics (SPH) is limited, drawing from our extensive experience with Material Point Method (MPM), we can attest that accurately simulating fluid flows using MPM can be highly challenging. It often necessitates supplementary numerical techniques, workarounds, and adjustments to ensure accuracy, primarily due to the inherent instability of MPM.

In contrast, FVM/CFD is a widely accepted and robust method, rooted in strong theoretical foundations. It doesn't require extensive "hacking" or unconventional code adjustments to yield reliable results. In essence, the choice between Eulerian and Lagrangian methods reflects a nuanced evaluation of trade-offs, where FVM/CFD stands out as a proven method with the ability to effectively capture complex fluid dynamics without the need for extensive ad hoc solutions.

4. Please explain the difference between ICE and general FVM/CFD. Baumgarten et al. (2021) has done a similar work by coupling FVM and MPM. The authors stated that their improvement is using implicit time integration for the multi-phase flows instead of explicit time integration for the single-phase flow. So what are the challenges when handling multi-phases flows with an implicit integrator? Multi-phase flows have been extensively studies in the fluid community, so the authors need to make their contribution more clear.

ICE (Implicit Continuous-Fluid Eulerian) is a subdivision of the Finite Volume Method, offering certain advantages in comparison to other approaches in the realm of flow computation encompassing all velocity ranges. The central concept behind this method is the utilization of a semi-implicit time discretization. This approach involves treating acoustic waves with an implicit scheme while handling advection terms explicitly. This dual approach enables the ICE

method to circumvent the Courant stability constraint, which typically relies on the speed of sound within the fluid. According to Harlow and Amsden [1], this method stands out as both numerically stable and efficient when addressing transient and viscous fluid flows in multiple spatial dimensions.

We just want to clarify that:

"Baumgarten et al. \cite{Baumgarten2021} made the first attempt at coupling the FVM with the MPM for the simulation of soil-fluid interaction by using an explicit time integration for the single-phase flow. In contrast to the mentioned work, we use implicit time integration for the multi-phase flows."

The challenge when handling multi phases flows is that complex Interfaces: Interfaces between different phases can be highly complex and subject to continuous changes. Capturing these accurately within the implicit framework can be demanding and may require advanced techniques like level-set or volume-of-fluid methods. We adopted the later one.

5. The multi-phase flows are considered in this work, but the authors only implemented the Terzaghi's effective stress for saturated soils. How about the unsaturated case? Because the water and air are involved in the framework.

You're correct in pointing out that the current model focuses on saturated soils and Terzaghi's effective stress analysis. Regarding the issue of submarine landslides, it's important to note that in most cases, the soil is fully saturated. Thus, our emphasis remains on the saturated case to keep the analysis manageable and focused.

The inclusion of unsaturated soils, where water and air are both present, introduces additional complexities that require more advanced physical models and numerical treatments to be accurately represented. Simulating unsaturated soils demands consideration of factors like air-water interaction, capillary forces, and moisture content effects, which significantly expand the complexity of the model.

Given the existing limitations and complexities associated with unsaturated soil modeling, we only have focused on the saturated soil scenario where Terzaghi's effective stress analysis provides a practical framework. Implementing unsaturated soil behavior would require a comprehensive approach involving sophisticated constitutive models, consideration of hysteresis effects, and specialized numerical methods to accurately capture the physics. While unsaturated soil behavior is indeed an important aspect to address, it is an avenue for further research and development beyond the scope of the current study.

6. A follow-up question of the previous comment: Why did the authors not split water and air in the conservation equations starting from Eq. (4). All these equations are for single-fluid phase only.

The reason for presenting a single conservation equation for both air and water in the text is to simplify the explanation and avoid redundancy, as they share the same conservation equation within our model. However, in the actual computation, we do solve two separate conservation equations—one for air and another for water. This approach allows us to accurately capture the distinct behaviors of each fluid material during simulations.

To elaborate further, our model is designed to handle various scenarios, including multiple fluid and solid materials. In cases where different fluid phases or solid materials exhibit unique behaviors, we ensure that separate conservation equations are solved to accurately represent their dynamics.

7. Please give a reference properly to the MC model because a detailed implementation is not presented yet.

We provide the numerical implementation for MC model in the manuscript as below:

The numerical implementation follows the approach described in Clausen et al. \cite{Clausen}.

8. References are missing for the implementation of the turbulent model.

We have now provided the references for the Smagorinsky model.

10. Did the authors consider energy dissipation given by Eq. (22) into the energy balance conversation?

Yes, we did consider the energy dissipation in the frictional contact and that term is considered in equation (55) during numerical implementation.

11. What are T_s and T_f ? Temperature?

T_s stands for solid temperatures and T_f stands for fluid temperature.

12. Is the Darcy law in Eq. (30) applicable to the turbulent flow?

Certainly, the Darcy equation is not applicable to turbulent flows. Equation (30) is provided to estimate an approximate Darcy permeability (or hydraulic conductivity) for the purpose of validating analytical solutions where turbulent flow is not present. In all our simulations, we employ a nonlinear drag force relationship that takes into account the Reynolds number as an input parameter to accurately represent the behavior in turbulent flow scenarios.

13. The authors consider heat transfer but only through the solid phase, e.g., Eq. (55). Is that necessary to consider heat transfer for the submarine land sliding triggered by earthquakes?

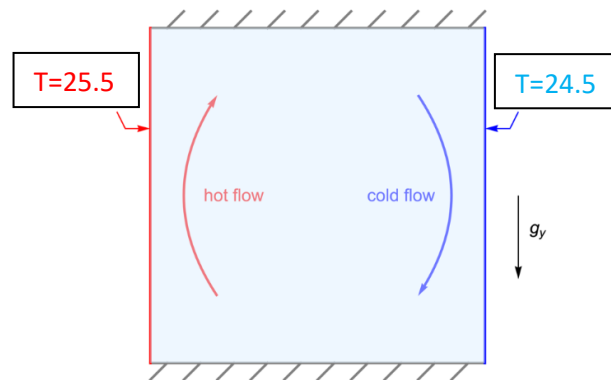
There is a potential that thermal effects could have an impact on landslides (Pinyol et al 2018), we have taken a cautious approach in our simulations. This is due to uncertainties in temperature boundary conditions. Consequently, we have set a constant temperature across all boundaries to minimize the potential thermal effects on our results.

N. Pinyol, M. Alvarado, E. Alonso, and F. Zabala. Thermal effects in landslide mobility. *Géotechnique*, 68(6):528–545, 2017.

It seems that the authors have also considered heat transfer through the fluid phase, but the formulations and implementations on convection are missing.

You're correct in noting that we have indeed considered heat transfer through the fluid phase. The implementation of energy exchange and convection is accounted for in the equations presented. Specifically for the numerical implementation, the convection of internal energy is addressed in Equation 79, and the overall energy exchange is accounted for by solving Equation 69 with linear system in line 420.

For the case "thermal induced cavity flow", why is it a clockwise flow because you have a symmetric boundary conditions? Moreover, is that possible to examined the transient response in addition to steady states?



The clockwise flow in the case of "thermal induced cavity flow" is due to the gravitational effect and temperature difference in the boundary. The gravitational force causes a density driven downward motion of the fluid on the cold side, and vice versa the hot fluid rises on left side, creating a clockwise rotation.

Regarding the examination of transient responses, yes, it is indeed possible to explore transient behavior. However, in our study, we have primarily focused on steady-state analyses. The simulation converged very quickly to steady states. Due to the length constraints of the paper, we chose not to present these transient responses in detail.

14. Is the rigid bed in Fig. 11 modeled by rigid material points or just a rigid body? Any damping involved in the simulation? How does the temperature evolve because you have considered the heat transfer? It might be negligibly small, right?

In Fig. 11, the rigid bed is modeled using rigid material points. As for damping, there's no damping involved in the simulation. I've made sure to clarify this aspect more comprehensively in the manuscript to avoid any confusion.

Regarding temperature, you're right. I consider its effect to be negligibly small on the simulation of earthquake-induced landslides. I've set the temperature to remain constant at the boundary. This approach effectively limits heat conduction's influence in the simulation.