Review: faSavageHutterFOAM 1.0: Depth-integrated simulation of dense snow avalanches on natural terrain with OpenFOAM

By Julia Kowalski

The manuscript *faSavageHutterFOAM 1.0: Depth-integrated simulation of dense snow avalanches on natural terrain with OpenFOAM* is a well-written contribution to the field of modeling and simulation of dense snow avalanches and fits into the scope of the journal. It extends the description of the OpenFOAM implementation first published by one of the authors, Matthias Rauter in (Rauter and Tukovic, 2018), to realistic topographies. The authors introduce the OpenFOAM specific workflow with a special focus on mesh generation from DEM data and GIS compatible post-processing. They demonstrate the capability of the proposed solver while analyzing a specific case study, namely the Wolfsgruben avalanche. Simulation results of the new OpenFOAM based solver are compared to the established tool samosAT.

My overall impression of this article is very positive due to the following reasons:

- First of all, by implementing the mathematical model into the mature software framework OpenFOAM, the authors actually outsource a lot of issues associated with e.g. data structure, parallelization and linear solvers. This allows a focus on the mathematical/physical model formulation itself, which is in my opinion the correct way to go.
- The mathematical/physical model itself is passed over to the solver in an encapsulated way as demonstrated on page 6 line 20 - 30. I find this a very big advantage of the approach. In fact this seems to be so straight forward, that one of the other reviewers even objected on the relative ease with which the model can be applied to real hazard scenarios, and the responsibility that arises from that. I certainly do have a slightly different opinion: While we as modelers do indeed have the responsibility to investigate and clearly communicate any limitations to the model, I fully support the authors approach in aiming towards modularizing software development and mathematical model representation as far as possible.
- Another aspect of the paper that I like very much is the comparison to an alternative simulation model. Although I have to admit that personally I find SamosAT a somewhat surprising choice (see my comment further down), this in principle is exactly the kind of results that help to better assess scope, potential and limitation of the various software tools that are in use.
- Finally, I want to emphasize the author's attempt to publish reproducible results. The paper refers to the code, which can be downloaded and tested.

All in all, I am clearly in favor of publishing this paper and think it will be a valuable contribution to the field, if the following objections have been addressed.

- Title seems odd: Probably the title is chosen according to the OpenFOAM module - still I think it doesn't reflect the content of the paper. While the mechanical model that the authors solve is similar to the Savage-Hutter model (see also my next comment), some of the original Savage Hutter theory's defining flavors, e.g. its questionable active-passive earth-pressure term and the 'dry-friction-only' resistive force, are absent. This , however, is also OK, as it does not seem to be what the authors want to promote, compare and discuss in their paper. Straight-on question: Why not faavalancheFOAM, for example?

- I like how you distinguish between the 'mechanical model' on the one side and the 'process model' on the other side. Personally, I would refer to it as the kinematic description and the dynamic closure, but it essentially means the same thing. Since it is important to differentiate between the two, I suggest to have the content of the footnote on page 2 integrated into the main text. As a side remark: I had to read the footnote several times, in order to digest and understand it. I think readability would benefit from an additional comma between 'integration' and 'and'. Or maybe rephrase into two sentences. Generally, I think it would be good to explain the name faSavageHutteFOAM (if you want to stick to it) based on the difference between mechanical and process model. It seems to me that the 'mechanical model' solved in this paper is similar to the Savage-Hutter mechanical model, while the 'process model' had been altered.

- Section 2.1: In the interest of the reader, it would be good to work over the structure of section 2.1.. Two concrete suggestions:
  1) Include a brief explaination of equations (1)-(3) just after they have been introduced rather than stating: Variables and mathematical operators are explained later .... I find it OK to refer to Rauter and Tukovic (2018) for details, but it must be possible to understand the paper without. In that context: Is there is typo in the last term of equation (2), what's the 'e'?
  2) Why not taking up the earlier thought ('mechanical model' vs 'model closure') in the structure of section 2.1, e.g a subsection mechanical model and its solution (description of the numerics, etc.), and a subsection on model closures , e.g. equations (9) - (12).

- Section 2.1: page 6 equation (11). Is z here the actual altitude rather than vertical coordinate? If so please make this more explicit.

- Section 2.1: page 6 line 20-30: I like including a code snippet. Given that the finite area aspect is new to most readers it would be very instructive to see a code snippet that has either a tangential or a normal operator in it. Does OpenFOAM have different operators for this? Or do you specifically implement their definition into the code?

- Section 2.2: page 6 line 14: 'Numerical diffusivity' does not 'prevent' oscillations! Diffusive behavior is rather a feature of first-order methods, while oscillations and lower diffusivity is a feature of second order methods. Please adjust the sentence accordingly.

- The authors talk about the model to work in 'moderately curved' topographies several times. Is it possible to quantify this somehow? What curvature values can be dealt with and what values are critical and can't be dealt with? Or from the perspective of a potential user of the code: Can you put this rather fuzzy observation in a concrete guideline and provide information for which cases it is not possible to use the code? If this is not possible, at least lay out what would have to be done to find such thresholds?

- Section 2.3: I like the description of the opensource workflow. A simple question that I could imagine some readers (inlcuding me) would be interested in: Is it also possible to run the OpenFOAM Solver with any arbitrary unstructured polygonial surface mesh? If yes, in which format does it have to come? Can you comment on this?

- Section 4: What the authors refer to as numerical uncertainty seems to be established in Roache 1997. Please repeat its definition here in the paper as it is unclear. It should be simply a measure of the accuracy of the numerical method, right? In that case the important aspect is that it can be controlled by the scheme itself (whereas typically other sources of uncertainty can't be easily controlled). This is why I don't like the expression 'numerical uncertainty too much, personally). I do disagree with your statement on page 15 around line 5, that the numerical implementation influences the results dramatically. I rather see a tendency that carefully implemented and quality assessed numerical schemes of different type solving the same mathematical model get closer and closer. What do you mean here exactly?

- Section 4: I like the comparison of the OpenFOAM results with results of other codes. To me it would however seem very valuable to compare against a model that is more similar in spirit, e.g. the Voellmy Salm portion of the r.avaflow project for instance, simply because it is also implemented as a finite volume based method, and the sources for differences could be tracked down more easily. Any reasons, why you chose the samosAT path instead?

All the best, Julia