Geosci. Model Dev. Discuss.,
https://doi.org/10.5194/gmd-2020-140-RC1, 2020

Geoscientific Model Development Discussions

Interactive comment on "*HydrothermalFoam* v1.0: a 3-D hydro-thermo-transport model for natural submarine hydrothermal systems" *by* Zhikui Guo et al.

Cyprien Soulaine (Referee)

cyprien.soulaine@gmail.com

Received and published: 14 August 2020

The paper "HydrothermalFoam v1.0: a 3-D hydro-thermo-transport model for natural submarine hydrothermal systems" describes the development of an open-source code based on the OpenFOAM C++ library for simulating hydrothermal systems using a Darcy model. Overall, the paper is well-organized and well-written. The code benefits from all the OpenFOAM features including parallel computing, three-dimensional domains, polyhedral grids. . . The validation cases presented in Section 5 are convincing and show the potential of such a toolbox.

The model implemented in HydrothermalFoam is very standard: a compressible single-

phase Darcy flow solver combined with a temperature equation. An important contribution is the integration of IAPWS-IF97 model in OpenFOAM. Although the manuscript does not bring any novelties in terms of modelling (everything is very well-established here), the implementation of the model into a modern and efficient platform can lead to significant progress in the submarine hydrothermal community. In particular, the authors have invested important efforts to make their code accessible to people that are not experts in OpenFOAM and the paper is written as a user manual. In that regard, I think the paper worth publications in Geoscientific Model Development.

I have noted some comments (see below) that should be checked before publication. Some are simply typos, others need to be checked carefully.

Comments:

l72-74: It is important to mention other initiatives that use OpenFOAM to solve flow and transport in porous media with Darcy-like solvers, e.g. Horgue et al. 2015 and Orgogozo et al. 2015

Horgue, P. et al., An open-source toolbox for multiphase flow in porous media, Computer Physics Communications, 2015, 187, 217-226

Orgogozo, L. et al., An open-source massively parallel solver for Richards equation: Mechanistic modelling of water fluxes at the watershed scale, Computer Physics Communications, 2014, 185, 3358 - 3371

l84: I think it is important here to mention that the PDEs are solved implicitly but sequentially.

l96: strictly speaking, in fluid mechanics, in laminar flow regime, there is inertia. Darcy's law corresponds to the creeping flow regime when inertia is negligible compared with viscous forces.

Eq (2) : the right-hand side (=0) is missing

l104: The compressibility of the rock is neglected as soon as the porosity is removed from the time derivative in Eq (1). Then, it is probably better to mention this hypothesis just after Eq (1).

the heat conductivity is noted lamba_r in Eqs (4)-(7) but k_r in the code.

I am not sure that the energy equation in Eq (4) (and then Eq (7)) is exact. It seems correct only if the continuity equation is a divergence-free equation which is not the case in the paper (this is what appears if you derive the equation from the conservation of the internal energy ddt(e) + div(e U) = div(q) + S with e(p,T)). Moreover, it is not clear whether the two last terms of the right-hand side are necessary as they seem orders of magnitude lower than the other terms. Can you comment on that? Give an estimation of the weight of these terms and explain in which cases it is important to consider them?

Assuming that the equation is correct, the numerical treatment presented in Listing 10 can be improved by making the last term implicit with fvm::Sp(alphaP*(...) , T), which should lead to better numerical stability.

l139: “adapt” instead of “adopt”

replaced all “can be” by “is/are”

#l159-160: the sentence “since it is not straightforward to impose fluid velocities on boundaries” is not clear. Actually, here you just transport a velocity value into a boundary condition on the pressure gradient because your solver solves a pressure equation only.

l166: what is the difference between submarinePressure and OpenFOAM’s PrghPressure?

#l177: more details will be appreciated on the implementation of the thermo-physical model.

#l194-1999: the sentence about constrainPressure and fixedFluxPressure is very confusing. Why do you need fixedFluxPressure in your simulations? What is the link with the boundary conditions introduced in Section 2.4?

#It is difficult for the potential user to know where is the code. It is scattered on too many platforms. The document mentions at least 3 different locations (Zenodo.org, DockerHub, GitLab). In particular, Zenodo.org and GitLab seem to provide the same code.

#l354-355, l364-365, l374-375: “can be be”, “bechmarks”

#l396 Zenodo.org

#Section 5: I think it will be interested to have a comparison of the simulation time of HYDROTHERM and HydrothermalFoam. Such a comparison can highlight better the importance of using modern computational platforms.

Interactive comment on Geosci. Model Dev. Discuss., https://doi.org/10.5194/gmd-2020-140, 2020.