PROGRAM SUMMARY OF PERMAFOAM

Program Title: permaFoam

CPC Library link to program files: (to be added by Technical Editor)

Developer's repository link: https://develop.openfoam.com/Community/hydrology/

Code Ocean capsule: (to be added by Technical Editor)

Licensing provisions: GPLv3

Programming language: C++

Nature of problem: This software solves the coupled equations that govern water flow and heat transfer in variably saturated and variably frozen porous media, for transient problems in Three-dimensional, heterogeneous domains. The equation for water flow is Richards equation, which is a very popular model for water transfer in variably saturated porous media (e.g.: soils), and the equation for heat transfer is a Fourier equation including advection and the freeze/thaw of the pore water. The solver is designed to take advantage of the massively parallel computing performance of OpenFOAM®. The goal is to be able to model natural hydrosystems of cold regions on large temporal and spatial scales.

Solution method: For each equation a mixed implicit (FVM for Finite Volume Method in the object oriented OpenFOAM framework) and explicit (FVC for Finite Volume Calculus in the object oriented OpenFOAM® framework) discretization with backward time scheme is embedded in an iterative linearization procedure (Picard algorithm). The coupling between the two equations is performed through an operator splitting approach. The implementation has been carried out with a concern for robustness and parallel efficiency.

Additional comments including restrictions and unusual features: This version of permaFoam has been tested with OpenFOAM_v1912 and OpenFOAM_v2106, thus everything might not work with other (especially older) versions of OpenFOAM. When using permaFoam, one should be careful to use fine enough spatial and temporal discretisations where and when steep fronts (freeze/thaw fronts, imbibition/drainage fronts) occure, otherwise numerical stability problems might arise.