

Home Faculty of Engineering Departments, institutes and centres
Department of Earth Science and Engineering Research Research Groups
Pore-Scale Modelling and Imaging Software Direct two phase flow solver

Direct two phase flow solver

The latest experimental version of the code based on the conservative curvature force formulation by [Shams et al \(2018\)](#) and the filtering of surface forces by Raeini et al. (2012) [[doi](#)] [[PDF](#)] is available from github:

<https://github.com/aliraeini/porefoam>

[See this Fishare link for the 2015 version of porefoam code](#) which is *based on* the algorithm published in Raeini et al. (2012) [[doi](#)] [[PDF](#)] with modifications to improve the stability of the code for unstructured meshes. It includes the required pre- and post-processing tools, sample input files and a short documentation. It has been used to study pore-scale events such as snap-off and piston-like displacement (Raeini et al. 2014a) [[DOI](#)] [[PDF](#)], to upscale pore-scale forces in two-phase flow (Raeini et al. 2014b) [[DOI](#)] [[PDF](#)] and to study capillary trapping on micro-CT images of porous media (Raeini et al. 2015) [[DOI](#)].

An older [version \(2011-2012\)](#) of the direct two-phase flow solver used for modelling two-phase flow at the pore/micron scale can be [downloaded here](#). It is closer to the algorithm published in Raeini et al. (2012) [[DOI](#)] [[PDF](#)] than the links above. The code should be linked to OpenFOAM; it has been tested to work with OpenFOAM version 1.6-ext, please see [OpenFOAM-extend website](#) or [its SourceForge repository](#).

For any enquiry, please contact Ali Raeini: a.gaseminejad-raeini09@imperial.ac.uk.