Cite as: Vergassola, M.: A continuous forcing immersed boundary approach to solve the VARANS equations in a volumetric porous region. In Proceedings of CFD with OpenSource Software, 2021, Edited by Nilsson. H., http://dx.doi.org/10.17196/OS_CFD#YEAR_2021

CFD with OpenSource software

A course at Chalmers University of Technology
Taught by Håkan Nilsson

A continuous forcing immersed boundary approach to solve the VARANS equations in a volumetric porous region

Developed for OpenFOAM-v2006
Requires: PyFoam

Author:
Marco Vergassola
Delft university of technology
M.Vergassola@tudelft.nl

Peer reviewed by:
FirstName1 LastName1
FirstName2 LastName2

December 17, 2021

Study questions

1. Which is an important difference between the porous models implemented in `olaFlow` and `OpenFOAM`? Why is this difference important?

   The porosity formulation implemented in `olaFlow` differs from the standard porous region of `OpenFOAM` as it keeps the porosity inside the differential operators in the momentum equation. This step is important whenever multiple porous layers with different characteristics are present as it ensures that fluxes across the interfaces are accurately computed.

2. Which type of immersed boundary method is used in the `porousPimpleIbFoam` and why?

   The continuous forcing approach is implemented. This is preferred over the discrete one because of its simplicity and lack of mass conservation problems at the IB. Moreover, the discrete approach implemented in `foam-extend` does not fit well with the purpose of simulating porous objects since it does not resolve the flow inside the body. Finally, the formulation of the porous drag force used in this project works well with the approach of a forcing term included in the governing equations before the discretization step.

3. How is the porosity field corrected at the immersed boundary? And why isn't there a porosity field for the solid region and one for the fluid region?

   The porosity field, n, is corrected with a VOF-like approach in which a scalar field called γ is defined to adjust the porosity level at the IB. Both the n field and the γ field are bounded between 0 and 1. Given this, and considering that outside of the IB $n = 1$, the corrected porosity, n_{corr}, can be written as $n_{corr} = \gamma + n\,(1-\gamma)$ and there is no need to define two n fields.