The 8th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2022
5-9 June 2022, Oslo, Norway

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

M. Vergassola[1], O. Colomés[1]

[1] Faculty of Civil Engineering and Geosciences, Delft University of Technology, Stevinweg 1, 2628CN Delft, Netherlands, M.Vergassola@tudelft.nl

Keywords: *Porous structures, Immersed Boundary Method, OpenFOAM*

The study of the interaction between waves and porous structures is of great importance in offshore engineering. These are often used to dissipate the energy of incoming waves and protect structures or land. For this reason, with time a few porous models have been implemented in CFD solvers, mostly focused on the simulation of large fixed porous structures, see [1]. These models represent an alternative to a microscopic representation of the porous region. In fact, often only the macroscopic effects, e.g. the total hydrodynamic force on the structure, are of interest. In this cases, a porous model can be used to model the flow-structure interaction, reducing the computational burden.

Currently, in OpenFOAM®, it is only possible to define a static porous region. This derives from the main use of these models. However, for the alternative applications aforementioned, it would be useful to allow the definition of dynamic regions. This would, for example, enable the use of porous regions for the study of Fluid-Structure-Interaction (FSI) problems involving perforated or porous objects. Generally speaking, in OpenFOAM®, a volumetric porous region is introduced with a similar approach to what is done for the Immersed Boundary Method (IBM) in its continuous forcing formulation. In fact, for this type of IBM, a set of cells are marked to track the location of a fictitious solid within the fluid mesh. Then, a forcing term is introduced in the momentum equation to simulate the presence of the body.

In this work, a new solver is developed to solve the Volume-Averaged Reynolds-Average Navier-Stokes (VARANS) equations inside a volumetric porous regions using a continuous forcing IBM. Compared to the porous media implemented in OpenFOAM®, this does not require a conformal mesh and it allows the definition of dynamic porous regions. The use of a simple Cartesian mesh represents an important advantage also when dealing with complex geometries as it eliminates any meshing problem. The solver is tested by simulating a static porous cylinder in a 2-dimensional constant flow at both Re=40 and Re=100. Both quantitative and qualitative results are promising. More tests are currently being performed. So far, the solver presents a speedup value of 1.88 compared to traditional conformal mesh solvers.

REFERENCES

[1] P. Higuera, J. L. Lara, and I. J. Losada, “Three-dimensional interaction of waves and porouscoastal structures using OpenFOAM®. part i: Formulation and validation”, *Coastal Engineering*, vol. 83, pp. 243–258, 2014.