# OpenFOAM implementation of immersed boundary method to simulate fluid-solid interaction problems

**†Magu Raam Prasaad R*[1], Ajit Kumar[2], and Santanu Mitra[1]**

[1]School of Engineering, Department of Mechanical Engineering, Shiv Nadar University, India
[2]School of Natural Sciences, Department of Mathematics, Shiv Nadar University, India

*Presenting author: rp361@snu.edu.in
†Corresponding author: rp361@snu.edu.in

## Abstract

In this work, we have implemented the well established immersed boundary method in OpenFOAM to simulate fluid-solid interaction of an incompressible viscoelastic solid immersed in an incompressible viscous fluid. We use the mathematical framework of the IB method to write down the momentum and continuity equation in a Eulerian form and compute the elastic force using a Lagrangian approach. The delta function is used to transform variables between the reference and the spatial domain. To implement the IB method in OpenFOAM, we discretize the spatial domain using a structured finite volume mesh and the reference domain using the tetrahedral mesh. We use the cell centered finite-volume schemes of OpenFOAM to discretize the momentum equation on a structured grid and an energy-based approach to compute the nodal forces. A smoothed version of the delta function is used to spread forces from moving Lagrangian nodes to the cell centers of the finite volume mesh. Similarly, we use the delta function to interpolate the velocities from cell centers to the moving Lagrangian nodes. We temporally discretize the IB equations in the following sequence, the elastic forces are computed at the beginning of the timestep and spread on to the finite volume mesh. To solve for velocity and pressure the incompressible Navier-Stokes equation is integrated using a semi-implicit fractional step projection method. To complete the time step the Lagrangian nodes are deformed to a new location by interpolating the velocity field. Test examples of fluid-solid interaction of hyperelastic solids are simulated in OpenFOAM and validated against the benchmark numerical studies.

**Keywords:** Immersed boundary method, fluid-solid interaction, OpenFOAM, finite-volume method.