Two phase flow simulations based on Immersed boundary method, by utilizing MUMPS solver

Avihai Spizzichino[1,2], Sharone Goldring[2] and Yuri Feldman[1]
[1] Department of Mechanical Engineering, Ben-Gurion University of the Negev
[2]Nuclear Research Center – Soreq

Multiphase flow is ubiquitous in our daily life and can be met in a large number of engineering applications. In many of them and especially in production process of fused optical fiber components, an accurate prediction of the two-phase flow characteristics by numerical simulations is of a great importance. For such components, the high fidelity numerical simulations allow for reducing research and development times, and for increasing production quality.

Numerical methodology allowing for an accurate prediction of the instantaneous shape of coupler during the fusing process was developed. Presently developed method is extremely efficient for 2D and axi-symmetric simulations. The developed code includes solution of incompressible viscous two-phase immiscible flow by utilizing interface tracking approach based on the immersed boundary method. This method supports two different grids: structured Eulerian grid, on which the pressure and the velocity fields are discretized by the finite volume method, and Lagrangian grid, reflecting the instantaneous position of the two-phase interface. The data transfer between both grids is implemented by two adjoint operators: interpolation operator transferring the Eulerian pressure and velocity fields to the Lagrangian points on the interface and regularization operator smearing the Lagrangian surface tension forces to the Eulerian grid in the vicinity of the interface. The principle novelty of our approach is to treat the surface tension forces as Lagrange multipliers enforcing the immiscibility of both phases. As a result, the whole system of continuity and momentum equations is augmented by additional constraints stemming from the surface tension forces applied at the two-phase interface. The assembled operator comprises indefinite large sparse matrix, which is efficiently solved by the direct sparse solver MUMPS at each time step. Such implicit formulation provides an accurate mass conservation of both phases without the need for additional correction steps.
The numerical simulations of two types were performed: first, the solution of benchmark problems to verify the recently developed code; second, the simulations aimed at comparison between the experimental and the numerical results obtained for fusion of optic fibers.
All the numerical results demonstrate an acceptable agreement (both qualitative and quantitative) with the previously published numerical and experimental data.
In summary, the performed study comprises an important milestone in developing fundamental hydrodynamic model for robust structural simulation of two phase flow, and fused fiber components