VI International Conference on Computational Methods for Coupled Problems in Science and Engineering
COUPLED PROBLEMS 2015
B. Schrefler, E. Oñate and M. Papadrakakis (Eds)

A high-order fully coupled Electro-Fluid-Dynamics (EFD) solver for multi-phase flow simulations

N. Emamy*, M. Karcher, R. Mousavi and M. Oberlack
Chair of Fluid Dynamics, Technical University of Darmstadt, Germany
*Correspondent author's email: n.emamy@fdy.tu-darmstadt.de

Numerical simulation of deformation of a droplet in a stationary electric field is performed in the present study. The droplet is suspended in another immiscible fluid with the same density and viscosity but a different dielectric property (permittivity). By applying the electric field, the fluids are polarized. The polarization process is accompanied by mechanical forces and deformation. A two-way coupling occurs because of the forces exerted from the electric field on the droplet and the deformation of the droplet which changes the geometry for the electric field computation. The droplet continues to deform until a force balance between the electric force, pressure and the surface tension is achieved and the droplet becomes a spheroid.

An electromechanical approach is adopted to solve the above mentioned problem, which includes solving the governing equations of both the electric and fluid fields, computing the coupling forces and capturing the movement of the interface of the droplet and the surrounding fluid. A one-fluid approach is followed, which enables us to solve one set of the governing equations for both the droplet and the surrounding fluid. The interface is represented as the zero iso-value of a level set function and an advection equation is solved to find the movement of the interface. A diffuse interface model is used to regularize the jump in the fluid and electric properties.

The governing equations of the electric and fluid fields and the level set advection equation are discretized using the discontinuous Galerkin Finite Element method (DG) using an in-house code for solving conservation laws. The electric field is computed from the electric potential by considering the electrostatics equations. To find the electric potential, a Laplace equation is solved which has a jump in the permittivity at the interface. The Laplace equation is discretized using the interior penalty method (IP), which we modified for the case of high jumps in the permittivity. Assuming that the fluids are linear dielectric materials, the electric force is the dielectrophoretic force. This force is added as a body force to the incompressible Navier-Stokes equations, which are the governing equations for the fluid flow. Considering that there is no jump in the fluid properties, a single phase solver of the Navier-Stokes equations including the surface tension at the interface is developed. The surface tension force is added as a body force to the Navier-Stokes equations, using the continuum surface force model (CSF). This model is known for producing a spurious velocity field. To decrease the spurious velocities, the surface tension term is calculated by using high degree polynomials for a precise calculation of the normal vector and curvature.

To solve the incompressible Navier-Stokes equations using the DG method, a projection scheme with a consistent Neumann pressure boundary condition is employed and the same polynomial degree for the velocity and pressure (equal-order method) is applied. However, using the DG method, we have observed that discontinuities in the solutions at the cell boundaries can affect the solution accuracy and even cause a numerical instability. These accuracy and stability issues occur when the derivatives of the solution are computed. Therefore a flux-based method for calculation of the derivatives of the flow variables and electric field was adopted. The simulations are performed for cases of steady and oscillatory electric fields, which predict the physical behavior expected.