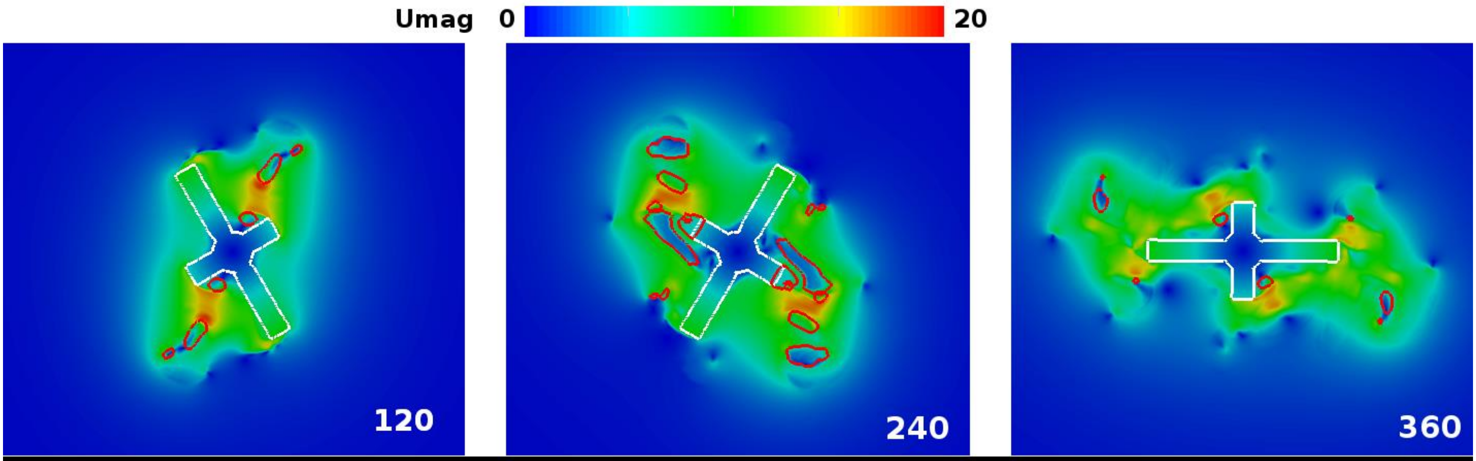
Summary of Deliverable D3.6: Immersed Boundary with Cavitation

Provided by ESR 10, Mr Evangelos Stavropoulos Vasilakis, under the supervision of Prof. Manolis Gavaises, CITY University of London, April 2017



Cavitation induction by rotating cross in water at rest. Velocity Magnitude field, at different roation angles, with 1% vapour volume fraction iso-line (red).

Within the framework of Computational Fluid Dynamics, simulations of flows of industrial interest often refer to complex geometries. In order to discretize the computational domain, the conventional strategy is to generate a geometry conformal grid, which poses challenges related to the complexity of the topology. When the problem includes moving parts, additional issues arise regarding the adaptation of the grid as the boundaries of the domain transform. Several methodologies are proposed for such problems, one of which is the *Immersed Boundary Method.*

Using the *Immersed Boundary Method* [1], the computational grid does not adapt to the geometry, but the presence of its boundaries is modelled through alternations of the governing equations. These alternations may be as simple as the addition of a source term in the equations or involve more complicated fluxes manipulation. The equations therefore can be solved on a simple static canonical cartesian grid, resulting in simpler mesh generation, smoother grids, lower computational cost, as well as easier domain transformation for problems with moving geometries. These methodologies have been extensively studied and applied in various problems, from biological flows to aeronautics and fluid structure interaction.

In the present study, a discrete forcing immersed boundary (IB) method, for simulating cavitating flows with moving boundaries, is presented. The method developed aims to facilitate the simulation of cavitating flows in complicated geometries and problems with moving parts, with main objective to be used to study cavitation induction during the operation of mechanical heart valves (MHV). The method, developed within the framework of OpenFOAM library, consists of introducing a source term in the momentum equation, which tries to lead the velocities of the cells covered by the immersed boundary to match the velocity of the boundary, imposing no-slip boundary condition on the boundary.

The developed method is tested, using a barotropic solver, on the cavitating cases of a shock tube and a simplified rotating geometry and validated against results from the literature [2] and simulations with conventional techniques. It is found that it offers a sustainable alternative to conformal grid methodologies, regarding cavitating flows with moving geometries. It preserves the physics, accurately captures the overall flow and simplifies the simulation of cavitation induction due to boundary motion. Problems with more complex geometries will be studied, in order to assess and enhance the performance of the method, taking solid steps towards the investigation of cavitation induction in Mechanical Heart Valves.

References

1. R. Mittal and G. Iaccarino, Immersed Boundary Methods, Annual Review of Fluid Mechanics, vol. 37, pp. 239-261, 2005
2. F. Örley, V. Pasquariello, S. Hickel and N. A. Adams, Cut-element based immersed boundary method for moving geometries in compressible liquid flows with cavitation, J. Comput. Physics, vol. 283, pp. 1-22, 2015