Water Flow Simulation with Computational Fluid Dynamics (CFD): a Review Study

Atheer Saleem Almawla[1], Assim Mohammed Lateef[2], Ammar Hatem Kamel[1]

Abstract – *Computational Fluid Dynamics (CFD) provides a numerical approximation to solve the governing equation of fluid movement. Four stages are required to analyse fluid problems through CFD techniques. The first one is to write the mathematical equation that should describe the fluid flow, which is set as a partial differential equation. The second stage is the discretisation of fluid flow mathematical equations in order to build the equations numerical analogue. The third stage is to divide the domain of fluid into small elements or cells. Finally, a specific problem equation is solved using the initial and the boundary conditions. The solver of problems usually is using the ether Finite Volume Method (FVM), the Finite Element Method (FEM), or the Finite Difference Method (FDM). The technical approach in this study provides students, researchers, and engineer a quick, comprehensive, and up-to-date tool and reference on the fundamentals, governing equation, and turbulence model applied in Computational fluid dynamics.* **Copyright © 2022 Praise Worthy Prize S.r.l. - All rights reserved.**