3D Steady and Transient Centrifugal Pump Flow within Ansys CFX and OpenFOAM

Authors : Clement Leroy, Guillaume Boitel

Abstract : This paper presents a comparative benchmarking review of a steady and transient three-dimensional (3D) flow computations in centrifugal pump using commercial (AnsysCFX) and open source (OpenFOAM) computational fluid dynamics (CFD) software. In centrifugal rotor-dynamic pump, the fluid enters in the impeller along to the rotating axis to be accelerated in order to increase the pressure, flowing radially outward into another stage, vaned diffuser or volute casing, from where it finally exits into a downstream pipe. Simulations are carried out at the best efficiency point (BEP) and part load, for single-phase flow with several turbulence models. The results are compared with overall performance report from experimental data. The use of CFD technology in industry is still limited by the high computational costs, and even more by the high cost of commercial CFD software and high-performance computing (HPC) licenses. The main objectives of the present study are to define OpenFOAM methodology for high-quality 3D steady and transient turbomachinery CFD simulation to conduct a thorough time-accurate performance analysis. On the other hand a detailed comparisons between computational methods, features on latest Ansys release 18 and OpenFOAM is investigated to assess the accuracy and industrial applications of those solvers. Finally an automated connected workflow (IoT) for turbine blade applications is presented.