Flow Analysis of a Single-Blade Pump with OpenFOAM

Mikko Auvinen
Helsinki University of Technology, Department of Applied Mechanics
P.O. Box 4400, Fin-02015, Finland
Phone:+358-9-451 3949 mikko.auvinen@tkk.fi

Single-blade pumps constitute a special group among pumps, featuring geometry designed to tolerate large object penetration. Such pumps have not been analyzed thoroughly in literature and reliable CFD methodology is still under development due to the immanent difficulties in applying CFD to such complex systems. For multi-blade centrifugal compressors and pumps, quasi-steady (frozen-rotor) analysis methods are widely utilized, while in the case of single-blade pumps, quasi-steady analysis can only be applied to configurations where the pump volute has been replaced by a rotationally symmetric disc-volute. The true pump configurations necessitate computationally intensive time-accurate analysis, which requires a sliding plane interface between the rotating and stationary domains. To elevate the level of computational difficulty, the single-blade geometry generates a flow system, drastically dissimilar to conventional pumps, exhibiting complex secondary flow behavior and strong oscillations that arise from the impeller-volute interaction.

This study utilizes the recently introduced Generalized Grid Interface (GGI) to conduct a comprehensive flow analysis of an experimental single-blade pump design with the objective of resolving the complex flow behavior of the system and attaining accurate hydrodynamic performance measures. Since it is known that the non-periodic geometry requires high fidelity CFD analysis, great care has been invested in generating high quality computational grids for this study. The rotating and stationary domains of the pump have been separately created with *GridPro* mesh generation software and imported into OpenFOAM with the *GridPro2Foam* converter. The two domains were merged together with the *mergeMeshes* utility. The coupling between the two non-conformal cylindrical mesh patches at the interface of the two domains is achieved with the GGI.

The recently launched computationally intensive time-accurate analysis was initiated with a quasi-steady (frozen-rotor) solution obtained with the *MRFSimpleFoam* solver, which provided a reasonable, but non-physical, initial guess for the flow field. The unsteady Reynolds averaged Navier-Stokes (URANS) simulation is carried out with the *turbDyMFoam* solver, which utilizes the *mixerGgiFvMesh* library to handle the time dependent mesh motion. Due to the non-physicality of the quasi-steady initial solution, the simulation requires multiple complete revolutions to convect the faulty information out and settle to a recurring oscillatory pattern. Furthermore, in order to maintain a stable numerical behavior, the length of the time step has to be kept at a level which corresponds to a revolution of 0.5 degrees. Within this time step, a tight convergence of momentum residual is ensured with 5 outer iterations.

The obtained intermediate results (work is still in progress) demonstrate the expected oscillatory behavior the system, but currently (after 4 revolutions) the consecutive revolutions still indicate that the flow system continues to evolve. Once the flow system reaches a steady oscillatory pattern, the results will be compared to empirical performance and axial force measurements. The on-going project also entails empirical work for acquiring LDV velocity vector field data from the analyzed pump design. This data will be used to compare against computational results obtained with OpenFOAM. (Using the latest data, this comparison will be illustrated at the workshop.)

The presentation at the workshop will aim to demonstrate the associated CFD methodology used in the pump analysis – with a particular emphasis on OpenFOAM related issues, including geometry and grid handling aspects.