

Adapting OpenFOAM for Hydroturbines

Maryse Page , page.maryse@ireq.ca Martin Beaudoin , beaudoin.martin@ireq.ca	Hydro-Québec, Institut de recherche 1800 Lionel-Boulet, Varennes, J3X 1S1, Canada
--	--

Abstract

The availability of a high quality Open Source CFD simulation platform like OpenFOAM® offers new R&D opportunities by providing direct access to models and solvers implementation details. OpenFOAM well designed object-oriented C++ architecture allows us to efficiently enhance the toolbox by focusing on specific libraries and by re-using existing capabilities.

Developments have been made by Hydro-Québec to adapt OpenFOAM for hydroturbines [1]. We will describe the developments that have been made to implement these new turbomachinery related capabilities : multiple frame of reference solver (simpleTurboMFRFoam), domain coupling interfaces (ggi, cyclicGgi) [2][3], mixing plane interface and specific boundary conditions. Furthermore, we will discuss our on-going development related to high-performance computing for OpenFOAM. The development of these new capabilities and their validation have been made in collaboration with the OpenFOAM Turbomachinery Working Group [4][5][6]. Practical use of the turbomachinery capabilities will be shown for the analysis of a Francis hydraulic runner.

Key words: OpenFOAM Turbomachinery Working Group, Hydroturbine

References

- [1] M.Page and M.Beaudoin: *Adapting OpenFOAM for Turbomachinery Applications*, 2th OpenFOAM Workshop, Zagreb, Croatia, June 7-9, 2007.
- [2] M.Beaudoin and Hrvoje Jasak: *Adaptation of the General Grid Interface (GGI) for Turbomachinery Simulations with OpenFOAM*, 3rd OpenFOAM Workshop, Milano, Italy, July 10-11, 2008.
- [3] M.Beaudoin and Hrvoje Jasak: *Development of a Generalized Grid Interface for Turbomachinery simulations with OpenFOAM*, Open Source CFD International Conference, Berlin, Germany, December 4-5, 2008.
- [4] H. Nilsson, M.Page, M.Beaudoin, B.Gschaider and H.Jasak: *The OpenFOAM Turbomachinery Working Group, and conclusions from the Turbomachinery Session of the Third OpenFOAM Workshop*, IAHR, 24th Symposium on Hydraulic Machinery and Systems, Fos do Iguassu, Brazil, October 27-31, 2008.
- [5] O.Petit, M.Page, M.Beaudoin and H.Nilsson: *The ERCOFTAC centrifugal pump OpenFOAM case-study*, 3rd IAHR International Meeting of the Workgroup on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Brno, Czech Republic, October, 2009.
- [6] O.Petit, M.Page, M.Beaudoin and H.Nilsson: *The ERCOFTAC centrifugal pump OpenFOAM case-study*, 4th OpenFOAM Workshop, Montreal, Canada, June 1-4, 2009.