MESH CONVERGENCE STUDY WITH FOAM-EXTEND 3.1 FOR HYDRAULIC TURBINE DRAFT TUBE

DEVALS C[1], VU TC[2], ZHANG Y[3], DOMPIERRE J[4], GUIBAULT F[5]

[1]*École Polytechnique de Montréal,christophe.devals@polymtl.ca*

[2] *École Polytechnique de Montréal, thicvu@gmail.com*

[3]*École Polytechnique de Montréal, ying.zhang@ polymtl.ca*

[4]*École Polytechnique de Montréal, julien.dompierre@ polymtl.ca*

[5]*École Polytechnique de Montréal, francois.guibault@polymtl.ca*

Keywords: *mesh convergence, foam-extend 3.1, draft tube*

Computational flow analysis has become an essential tool for hydraulic turbine designers. The first step in the flow analysis process is to create a valid mesh for the computational flow domain and the CFD engineer has to figure out what is the desirable mesh size to get a mesh independent flow solution. But the accuracy of a RANS flow solution does not depend only on the mesh size. There are several parameters that influence the final solution, such as the distance of the first near wall point (y+ value), the mesh distance expansion factor toward the core flow, the size of the element in core flow for a given flow Reynolds number, element skewness and orthogonality, etc. Even though many studies have addressed the issue of mesh independence [1], there is still no definitive consensus on mesh best practices, and research on that topic is still needed.

This paper presents a mesh convergence study for turbulence flow in hydraulic turbine draft tubes as shown in Figure 1 which represents the most challenging turbine component for CFD predictions. Several parameters will be considered as the mesh distance expansion factor from solid wall which contribute to resolve the viscous flow boundary layer. Also the control of mesh size in core flow and their distribution in span wise and through flow directions will be taken into account. The draft tube flow analysis, through the parametric study, will be performed with simpleFoam (foam-extend-3.1 version) and the k-ε turbulence model for a wide range of inlet swirling flow representing turbine runner operating conditions from part load to full load. The numerical results will be compared with experimental data from hydraulic laboratory.

a) dtA b) dtB

Figure 1. Geometries

Acknowledgements

The authors would like to thank Andritz Hydro and the National Science and Engineering Research Council of Canada (NSERC) for its support of this research, under project RDCPJ 441752-12.

References

[1] R. Magnan, S. Cupillard, G. Gauthier, A.M. Giroux, M. Page and C. Deschênes, Challenges in assessing the grid sensitivity of hydro-turbine CFD simulations, 27th IAHR Symposium on Hydraulic Machinery and Systems (September 22-26, 2014. Montreal, Canada), 2014