CFD model of an aerating hydrofoil

D Scott¹, M Sabourin¹, S Beaulieu¹, B Papillon¹ and C Ellis²
1ALSTOM Renewable Power Canada Inc., 1350 ch. St-Roch, Sorel-Tracy, QC, CAN
2 St. Anthony Falls Laboratory, U. Minnesota, Minneapolis, MN, USA

E-mail: david.scott@alstom.com

Abstract. Improving water quality in the tailrace below hydroelectric dams has become a priority in many river systems. In warm climates, water drawn by the turbine from deep in a reservoir can be deficient in dissolved oxygen (DO), a critical element in maintaining a healthy aquatic ecosystem. Many different solutions have been proposed in order to increase the DO levels in turbine discharge, including: turbine aeration systems (adding air to the water through either the turbine hub, the periphery or through distributed aeration in the runner blades); bubble diffusers in the reservoir or in the tailrace; aerating weirs downstream of the dams; and surface water pumps in the reservoir near the dam. There is a significant potential to increase the effectiveness of these solutions by improving the way that oxygen is introduced into the water; better distributions of bubbles will result in better oxygen transfer. In the present study, a two-phase Computational Fluid Dynamics model has been formulated using a commercial code to study the distribution of air downstream of a simple aerating hydrofoil. The two-phase model uses the Eulerian-Eulerian approach. Appropriate relations are used to model the interphase forces, including the Grace drag force model, the Favre averaged drag force and the Sato enhanced eddy viscosity. The model is validated using experimental results obtained in the water tunnel at the University of Minnesota’s Saint Anthony Falls Laboratory. Results are obtained for water velocities between 5 and 10 m/s, air flow rates between 0.5 and 1.5 sL/min and for angles of attack between $0^\circ$ and $-8^\circ$. The results of this study show that the CFD model provides a good qualitative comparison to the experimental results by well predicting the wake location at the different flow rates and angles of attack used.

1. Introduction
Hydroelectric turbines are the world’s largest source of installed renewable energy, providing 16% of the electric demand. While hydroelectric energy is “green”, improvements can be made that will further reduce the ecological impact of this massive source of electricity. One factor in which hydroelectric turbines play a key role is in water quality; turbines do not have to be passive elements that simply transfer water from a reservoir to a tailrace; they can actively improve the quality of water flowing through a river, and they can do this while incurring relatively few additional hydraulic losses and at relatively low cost.

In warm climates, water drawn by a turbine from deep in a reservoir can be deficient in dissolved oxygen (DO), a critical element in maintaining a healthy aquatic ecosystem. Most common in warm climates, this problem occurs when bi-annual turnovers of the reservoir due to seasonal temperature variations do not occur; the reservoirs are often stratified with warm water on top and colder water at the bottom. Near the surface of the reservoir, oxygen levels are replenished through interaction with the atmosphere. Deep in the reservoir, however, sinking organic matter decomposes and consumes
oxygen, which can result in severe DO deficiencies near the depths at which turbines often draw water.

Instead of passively transporting this low DO water from the reservoir to the river, the turbine can actively be used to add oxygen to the water passing through it. By carefully designing the turbine to draw air into the hydraulic passages at naturally occurring low-pressure points (associated with higher velocity water), it is possible to mix air with water in the turbine and release high DO content water into the tailrace, all with little or no pumping costs. The goal is then to design such Auto Venting Turbines (AVTs) that have the least possible performance losses due to the addition of air to the flow and that transfer the maximum possible quantity of oxygen from the air into the water.

It is well known that the hydraulic losses in a turbine increase with the volume fraction of air in the water. Adding air to the water in order to increase the DO levels will have an impact on turbine performance, so it is necessary to find a balance between the amount of air injected in order to attain the required DO levels and the reduction in turbine efficiency. From data provided in March [1] and in Foust et al. [2] on an AVT at the Osage plant, it was calculated that the potential DO uptake for the quantity of air injected is significantly greater than is achieved. Overall, less than 50% of the oxygen in the air injected into the water is transferred as DO, the remainder leaves the water as the bubbles rise to the surface. A better understanding of the mechanisms of DO transfer will lead to better air injection systems that will reduce the amount of air required to increase DO levels. This will result in improvements in turbine efficiency due to reduced air volume fractions in the water.

Computational fluid dynamics (CFD) has become a key tool in the design and optimization of hydroelectric turbines. While the simulations of single-phase problems are well defined, two-phase simulations are still relatively new and are constantly under development. No general consensus exists on the best methods to use and almost every problem has its own particularities that require special attention. In the case of aerating turbines, two-phase fluid flow occurs as bubbles of air are injected into the water through passages in the turbine, either through the cone or through the runner.

In the present study, the two-phase flow modeling capabilities of a commercial CFD solver were used to simulate the flow of water around an aerating profile in a simple geometry. Validation of the numerical model was performed through comparison with experimental results obtained using a water tunnel at the University of Minnesota’s St. Anthony Falls Laboratory. This work is intended to provide a stepping-stone to the full two-phase analysis of an aerating turbine; the model is tested first on the relatively simple geometry of a NACA 0015 profile at small angles of attack in a uniform flow of water.

2. CFD model
The present study uses a commercial CFD solver to resolve the steady state equations of conservation of mass and momentum in a two-phase flow of air bubbles in water. The energy equation was not resolved, only mass, momentum and the interactions between the phases were considered. An Eulerian-Eulerian approach is used in which the air and the water are considered to be two interpenetrating fluids; the water is described as continuous, the air as dispersed. There is no transfer of mass from the air bubbles to the water in this work, only the air bubble distribution is investigated.

The turbulence in the continuous phase (water) is modeled using the k-ω Shear Stress Transport model. This model has the advantage of better predicting flow separation than a standard k-ε model, a feature that is particularly applicable to flows around hydrofoils. Turbulence in the dispersed phase (air bubbles) is accounted for using a dispersed phase zero equation (constant eddy viscosity Prandtl number throughout the domain).

Interactions between the two phases are taken account by the CFD code through the use of additional terms in the conservation equations. In this study, three main inter-phase terms are used, these being: interphase drag, interphase turbulent dispersion forces, and particle induced turbulence.

Interphase drag has been accounted for by use of the Grace drag force model [3]. This model was specifically developed for both dilute and dense bubbly flows; a user-specified exponent permits corrections for the density of the bubbles in the liquid to be accounted for. A value of 2 was used for
the calculations and was found to provide good results for relatively dense flows of 1mm diameter air bubbles.

Interphase turbulent dispersion forces are taken into account through the use of the Favre averaged drag model [4]. The default dispersion coefficient of 1 was used in the present work. This drag model was developed for flows of light dispersed phases in a heavier liquid. This is particularly applicable to the flow of air bubbles in water, as studied here.

Particle induced turbulence was included in the model in the form of the Sato enhanced eddy viscosity [5]. This interphase turbulence force affects the continuous phase eddy viscosity and accounts for the effects of the particles in the flow.

The dissolution of oxygen from air bubbles to the surrounding water is not instantaneous; it takes place over many seconds. To perform a complete simulation of the air bubbles injected into a turbine would require a domain that is prohibitively long (order of hundreds of metres) in order for the bubble residence time in the domain to be sufficiently long to determine the DO uptake to the water. For the present study, only a limited domain has been considered for which detailed experimental results are available; in fact, the calculation domain was selected to simulate an experimental water tunnel.

The domain for the CFD simulation is sketched in Fig. 1, showing the domain extents and the positioning of the profile. More details of the profile can be found in Fig. 2. The geometry was based on that of an experimental water tunnel in which the results used for validating the model were obtained. While the majority of calculations were performed in an essentially two-dimensional domain (0.5mm thick with a single volume in the x-direction), some calculations including a full three-dimensional geometry were also performed. In these cases, the depth of the calculation domain was 19.0 cm. In all cases, the top and bottom walls were defined as no-slip for both the air and water phases, a uniform velocity profile was applied at the inlet with water speeds between 5 and 10 m/s and no air in the water, and a constant pressure condition was applied to the outlet. The profile was defined as having no-slip walls with the exception of a 0.5 mm wide air injection slit on the bottom side. This air injection slit is defined as an inlet where air exits normally to give a constant volume flow rate between 0.5 and 1.5 sL/min (standard L/min: air flow normalized to the standard atmosphere). In the two-dimensional calculations, symmetry planes were defined on the sidewalls and the air injection slit covered the entire width of the calculation domain (0.5 mm). In the three dimensional calculations, the sides were defined as no-slip walls and the air injection slit was defined as a 1.0 cm wide slit centered in the domain.

Figure 1: Schematic illustration of the calculation domain used for CFD simulations.

A picture of the base mesh (referred to as V1) used in the calculations is shown in Fig. 3. This essentially hexa mesh, used for the simulations at 0° angle of attack, is finely refined near the profile surface. Additional refinement is used near the air injection location on the bottom surface in order to sufficiently discretize this surface.
Figure 2: Schematic illustration of the experimental NACA 0015 aerating profile used in this study showing key dimensions and the aeration slit on the bottom side.

The timescales used in the calculations were varied according to the amount of air injected into the domain. At lower air flow rates (0.5 sL/min), timescales of $10^{-2}$ s were sufficiently small to provide satisfactory results. At higher air flow rates (1.5 sL/min), it was necessary to significantly reduce the physical timescales used for relaxation to $10^{-4}$ s, significantly increasing calculation times. For larger timescales, only 300 to 500 iterations were required to attain converged solutions; for smaller timescales, it was necessary to use over 2000 iterations.

Figure 3: Close-up view around the aerating profile of the base mesh used at 0° angle of attack for the simulation of air injection from the profile into the stream of water (approximately 107 000 nodes).

3. Experimental Apparatus

Experimental results for validation of the numerical results were obtained in the University of Minnesota’s St. Anthony Falls Laboratory water tunnel. This water tunnel has a test section 19 cm wide by 19 cm deep by 120 cm long in which different hydrofoils can be placed. These profiles are equipped with an air injection system which draws air into slits on the profile. A PIV imaging system is used to track the bubbles in the flow downstream of the profile, permitting detailed measurements of both the instantaneous and mean bubble paths to be obtained. The bubbles themselves are used as the tracers for these measurements. More details on this tunnel can be found in Kjeldsen and Arndt [6]. Further information on the PIV system used can be found in Ellis et al. [7].

The profile used in the water tunnel was a specially designed NACA 0015 hydrofoil with a chord of 81 mm and a 0.5 mm wide air injection slit located on the bottom side near the leading edge (see
Fig. 2). Experiments were performed for angles of attack of 0°, -4°, and -8°, with mean freestream water velocities between 5.0 m/s and 10.0 m/s. Since the air bubbles used by the PIV system are significantly larger than tracking particles typically used, injection of air over a larger slot width resulted in the presence of an essentially opaque layer of air bubbles; the PIV system was unable to distinguish one bubble from another. Thus, for the tests in these experiments, air was injected only in a restricted section of the slit; tape was applied to the foil’s surface to mask all but the centre 5% (9.6 mm) of the air injection slot, permitting the PIV system to distinguish the bubbles. An added benefit of masking all but the centre portion of the slot was that it allowed the images to be calibrated to a known length scale. As a result, measured bubble sizes and velocities were accurate to +/- 3%. Under these conditions, the air flow rates used were 0.5, 1.0 and 1.5 sL/min. The air-to-water void fractions ranged from a minimum of 2.31e-5 to a maximum of 1.38e-4.

4. Results

4.1. Model validation and grid independence

The CFD model was initially validated by comparing single-phase (water only) results of the lift around a NACA0015 hydrofoil with reference data [8]. From this validation, performed for two different meshes, it was found that the variation of the lift coefficient with angle of attack calculated by the CFD model varied by no more than 5% from the reference values, and this at only one angle of attack (6°). For all other cases, the differences between the CFD calculated and reference values were less than 2.5%. Furthermore, these results, obtained for both a coarse and a fine mesh, confirm that the coarsest mesh may be used in the calculations; the results are essentially independent of the calculation grid used. These results are presented in Fig. 4.

![Figure 4: Experimental and CFD model calculated lift coefficients around a NACA0015 profile at different angles of attack for Re = 10^5.](image)

Further confirmation of the independence of the results on the mesh was obtained when looking at the velocity deficit in the wake region behind the profile for a freestream water velocity of 5.0 m/s. The velocity deficit region occurs where the z-direction velocity in the wake is less than the freestream value. To investigate the impact of the grid refinement on the velocity deficit, the variation in the CFD calculated z-direction velocities on a line 125 mm downstream from the leading edge of the
profiles was plotted for four different grids. As can be seen in Fig. 5, the differences between different grid refinements (V1 = coarsest – 107 000 nodes, V4 = finest, 320 000 nodes) were slight; the V1 mesh was used as the basis for further calculations and the results are considered to be essentially independent of the grid.

Figure 5: Variation of the velocity deficit in the wake region behind the NACA0015 profile at 0\(^{\circ}\) angle of attack and a freestream water velocity of 5.0 m/s for four different grid refinements.

4.2. Comparison with experimental results
The results from the two-dimensional CFD model were compared with experimental results obtained from the water tunnel at the University of Minnesota’s St. Anthony Falls Laboratory. The numerical calculation domain was defined in order to represent the geometry of this water tunnel, allowing direct comparisons of the CFD and experimental results.

Measurements of the flow in the water tunnel were performed using a PIV imaging technique. In these measurements, the air bubbles formed from the injection of air into the water were used as the tracer particles; a high-speed camera took pictures of the flow at different downstream locations and an analysis of the bubble motion allowed both instantaneous and time-averaged values of the velocity fields to be calculated. For this study, only the time-averaged results were used.

A comparison of the velocity contours for the case of 0\(^{\circ}\) angle of attack, 5 m/s freestream water velocity and 0.5 sL/min injected air is shown in Fig. 6, where windows showing the contours over the time-averaged z-velocity components calculated from the experimental measurements are inset over the numerically calculated values. It can be seen that the experimentally measured wake has almost completely dissipated before the end of the calculation domain while the numerically calculated value continues to dissipate.

A three dimensional analysis of the flow domain was performed in order to verify the 3D effects on the velocity deficit calculated by the CFD model. In this three-dimensional calculation, the side walls, located 19 cm apart, are given no-slip conditions, compared with the symmetry condition used in the two-dimensional analysis. Air is injected through a 1 cm wide slit in the center of the domain, as was performed in the experimental measurements. The numerical obtained z-direction velocity contours for the same conditions presented previously (angle of attack of 0\(^{\circ}\), a freestream water velocity of 5.0 m/s and 0.5 sL/min injected air) for this three-dimensional analysis are presented in Fig. 7, where the original two-dimensional results are plotted in (a) and the three-dimensional results in (b). The plane in which the contours are calculated corresponds to the middle of the 1.0 cm wide aeration slot from
Figure 6: Comparison of the numerical and experimental (inset) two dimensional contours of the variation of the axial velocity in the wake behind the aerating profile at an angle of attack of 0°, a water freestream velocity of 5.0 m/s and air injected at 0.5 sL/min.

Figure 7: Comparison of water velocities from (a) two-dimensional and (b) three-dimensional simulations of the water flow over the aerating profiles for 0° angle of attack, a freestream water velocity of 5 m/s and 0.5 sL/min injected air showing the improvements when solving the full three-dimensional domain.

The variations in the z-direction velocity at three locations located (a) 125, (b) 260 and (c) 393 mm downstream from the aerating profile leading edge are shown in Fig. 8. The velocity deficits predicted by CFD for both the two-dimensional and three-dimensional simulations are larger than those found in the experiments. The two-dimensional CFD predicted velocity deficits are typically 30% larger than the experimental values nearer the profile and the difference in the deficit increases to 60% further away. Also, the width of the numerical wake is significantly less than that predicted experimentally. This is most noticeable closer to the aerating profile where the predicted wake is only a third as large as the experimental wake. There is less of a difference towards the end of the calculation domain for the two-dimensional calculations. An improvement is noted in the three-dimensional CFD calculations; however the velocity deficit is still over-predicted by 25% in a line 125 mm from the leading edge and by 35% in a line 393 mm from the leading edge.

Overall, the two-dimensional simulations tended to overpredict the velocity deficit and underpredict the size of the wake with respect to the experimental measurements. A significant improvement was noted for the three-dimensional simulations.
The wake regions at a fixed location 393 mm downstream from the leading edge of the profile are plotted in Fig. 9 for four different flow conditions, comparing the experimentally and CFD determined wake regions. The conditions shown cover the extremes considered in this study; minimum and maximum values of water tunnel freestream velocities (5 and 10 m/s) and air flow rates (0.5 and 1.5 sL/min). The three different angles of attack used in the investigation are presented, showing results at angles of attack of a) 0°, b) -4° and c) -8°. The z-velocities were normalized by dividing by the freestream water velocity.

As was noted in Fig. 8, the results in Fig. 9 show that the two-dimensional CFD model overpredict the velocity deficit and underpredicts the width of the wake. However, the location of the center of the wake is well predicted by the two-dimensional CFD model. This is a key result, showing that the CFD model does a good job of predicting the location of the wake region in the flow, even if velocity deficits predicted by the CFD model do not correspond exactly to the experimental results.

5. Conclusions

Computational Fluid Dynamics is an extremely useful tool for modelling hydroelectric turbines. While single-phase CFD analyses are commonly used in turbine design, much work remains to be done on multi-phase analyses.

In this study, a two-phase CFD model of a flow of water around an aerating profile was performed. The results of this study showed that the two-dimensional CFD model well predicts the location of the wake region behind the profile, which is a key benefit to the use of CFD. Quantitatively, however, the two-dimensional model tends to overpredict the velocity deficit and underpredict the width of the
wake. Improvements in the comparison between calculated and experimental results were found when using a three-dimensional CFD model.

Ultimately, the goal of turbine modelling is to accurately describe the flow in through the hydraulic passages and to predict the turbine performance. Two-phase modelling of an aerating turbine is necessary in order to accurately predict the impact of the aeration on turbine performance, the dissolved oxygen uptake that can be obtained via different aerating techniques and to provide insights that will improve the DO transfer from the injected air to the water. The present work provides a stepping stone to this goal by demonstrating the validity of the proposed model of a two-phase, air-water flow over a simple geometry.

6. Acknowledgements
The authors would like to acknowledge the contributions of Mr. Mathieu Carignan, a student at the University of Sherbrooke, to the execution of the CFD models in this work during his internship at the Alstom Renewable Power Canada Inc. Global Technology Centre for Sustainable Hydroelectricity in the fall of 2013.

References
[1] March, P. (2011), “Hydraulic and environmental performance of aerating turbine technologies,”, EPRI Conference on Environmentally Enhanced Hydropower Turbines, Washington, D.C., May 19-21.
[2] Foust, J.M., Fisaher, R.K., Thompson, P.M., Ratliff, M.M. and March, P.A. (2009), “Integrating turbine rehabilitation and environmental technologies: aerating runners for water quality enhancement at Osage plant,”, Proceedings of Waterpower XVI, Spokane, Washington, USA, July 27-30.
[3] Grace, J.R. and Weber, M.E. (1982), “Hydrodynamics of drops and bubbles,” in Handbook of Multiphase Systems, ed. G. Hetsroni, Hemisphere.
[4] Burns, A.D., Frank, T., Hamill, I. And Shi, J-M. (2004), “The Favre averaged drag model for turbulent dispersion in Eulerian multi-phase flows,” paper #392, 5th Int. Conf. Multiphase Flow, ICMF ’04, Yokohama, Japan, May 30-June 4.
[5] Sato, Y. and Sekoguchi, K. (1975), “Liquid velocity distribution in two-phase bubbly flow,” Int. J. Multiphase Flow, Vol. 2, No. 1, pp. 79-95.
[6] Kjeldsen, M. and Arndt, R.E.A. (2008), “On similarities between flow systems operating at excessive gas load and at cavitation conditions: hydrofoil tests and CFD assessment,” Proceedings, 12th Int. Symposium on Transport Phenomena and Dynamics of Rotating Machinery, Honolulu, Hawaii, USA, February 17-22.
[7] Ellis, C., Karn, A., Hong, J., Scott, D., Gulliver, J. and Arndt, R. (2014), “Measurements in the wake of a ventilated hydrofoil: a step towards improved turbine aeration techniques,” Proceedings of the 27th IAHR Symposium on Hydraulic Machinery and Systems, Montreal, Canada, Sept. 22-26.
[8] Airfoil investigation database, Airfoildb.com.