THE BLOCKAGE EFFECT FOR INTAKE MEASUREMENT, A CFD APPROACH FOR CORRECTING THE DISCHARGE

GILLES PROULX
Hydro-Québec
5655 de Marseille
Montréal, Qc, H1N1J4
Canada
Proulx.Gilles@hydro.qc.ca

BERTRAND REEB
Électricité de France, D. T. G.
21 Avenue de l'Europe
Grenoble
France
Bertrand.REEB@edf.fr

ABSTRACT

Intake discharge measurements often require installing a fixed or movable frame supporting the instruments: a number of current meters and possibly pairs of acoustic scintillation (AS) transducers. The profiled bars or cylindrical piping often used in the design of movable frames raise questions regarding evaluation of the blockage effect correction in discharge calculations and the uncertainty associated with that correction.

The ISO 3354 test standard gives a relation for correcting discharge based on the ratio of the frontal area of the frame to the measurement cross-sectional area. Developed primarily for and suited to discharge measurements by the current meter method for a fixed frame, the correction formula may not be appropriate for measurements on a movable frame. Moreover, the AS method was ignored in deriving the formula and transducers are never calibrated on the frame. Two effects clearly influence velocity at the measurement instruments: increased velocity (blockage effect) due to the difference in the support frame blockage ratio during testing versus during calibration, and the local effect of the frame geometry where the transducers are located. The latter can have either a positive or negative effect on the measured velocity, depending on the frame design and transducer position on the frame. The local effect also changes as the frame moves along the measurement cross-section. The local effect exceeds the blockage effect under some site conditions.

CFD provides an appropriate approach for calculating the blockage effect under differing site conditions and for different frame designs, having been used for many years to evaluate aircraft performance, turbine design, etc. Also, the flow field geometry for intake discharge measurements is relatively simple and easy to manage with adequate test results so good simulations can be achieved in a reasonable amount of time with appropriate simplifications.

This paper studies two cases: measurement with a simple current meter frame and measurement with a frame supporting AS transducers. Results show a blockage effect closer to the lower limit in the ISO standard for the current meter frame. The effect on the discharge measurement for the AS method was much greater since the transducers were located close to the frame's main pipes.

1. INTRODUCTION

Measuring discharge at the intake of a low head power plant has many advantages over penstock measurement (Acoustic Transit Time, Current Meter, Pressure-Time) as it require no dewatering to install instruments and no unit downtime when performed using the maintenance gate slots and a moveable frame [4,**5**,6,7,8]. This method is also very economical for multi-unit power plants as the measurement instruments can be easily moved from one unit to another [1,3,9]. Though used mainly for low head power plants, the method can also be used at the intake of medium- and high-head power plants [10]. Another advantage is that measurements can sometimes be made without moving instruments, e.g., current meters at an intake to more than one unit [10].

There are only two common discharge measurement methods for low-head power plants: the current meter (CM) and acoustic scintillation (AS) methods. The CM method always requires a support frame and the AS method sometimes requires one. The blockage effect [10] caused by the frame must then be considered in the calculating discharge.

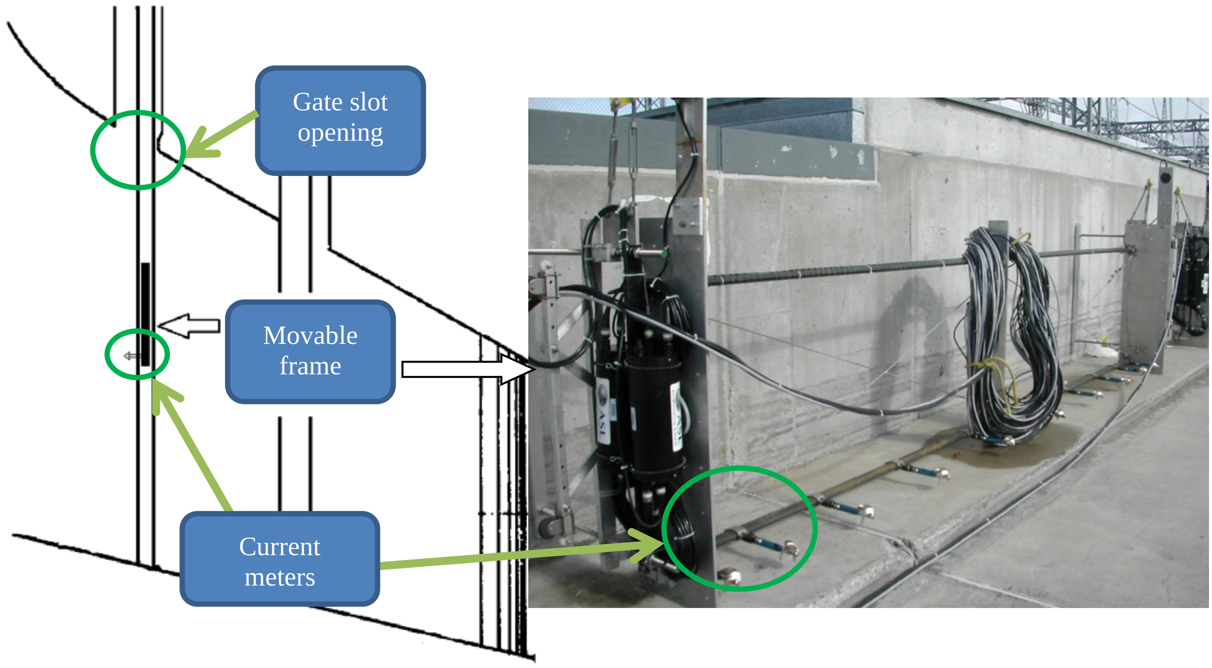

Figure 1: Typical intake profile and current meter frame used for discharge measurement at Hydro-Québec.

The conditions for flow rate measurements vary greatly from test to test. Designs range from slim frames (Figure 1) to robust ones made of large steel pipes. Flow angle and velocity distributions may vary significantly from site to site as well as in the measurement cross-section. This changing flow angle modifies the frontal area of the frame in almost every case. Most of the time, there is an opening (Figure 1) at the top of the measurement cross-section, which means that part of the frame could be out of the flow while the measurement instruments are in it. There, the proximity effect of the wall no longer influences measurements. Some frame designs support current meters on the upstream side far enough away to minimize the blockage effect.

The ISO 3354 [10] gives a small number for the blockage ratio calculated as the frontal area of the frame to the measurement cross-sectional area. However, the blockage effect is not evenly distributed and is partially moving with the current meters (proximity effect). The uncertainty given for the blockage effect correction is rather large with a value of ±2/3 the value of the correction itself. The blockage ratio is limited to 6 % but could easily be increased, which would be advantageous for taking measurements at a small power plant.

The current meters are calibrated in a towing tank with its own blockage effect. It is practically impossible to perform calibration with exactly the same blockage effect as in field tests. Meters are not normally calibrated prior to each test, as allowed by IEC 60041 [12]. In the towing tank, current meters are normally mounted on the same support rod but rarely extended to full depth so there is a 3D effect inexistent in the field. A large support frame is difficult to move in the towing tank, so it is rarely included during the calibration. In all cases, the rise in water level may significantly affect calibration accuracy. It may thus be concluded that the discharge must be increased (blockage effect greater in the calibration channel than in field tests) rather than lowered, as prescribed by the ISO standard.

For all of the reasons above, it is desirable to have a tool capable of calculating the real blockage effect under both site test conditions and calibration conditions. Computational fluid dynamics (CFD) simulation is such a tool. The type of measurement conditions that can be simulated are only limited by computational power and time available. Though simulations clearly will not be 100% accurate since most CFD software does not fully resolve Navier-Stoke equations, CFD simulations provide close agreement with test results in many fields, particularly for hydraulic turbine calculations.

The goal of the paper is to show that CFD can be used to calculate the blockage effect and correct the measured discharge. Two discharge measurement cases will be studied, one with the AS method and the other with the CM method.

2. METHODOLOGY

Many factors must be taken into account to perform reliable CFD simulations, including the meshing, especially the size of elements near the solid boundary, the turbulence model and the type of geometry (2D/3D).

The region of greatest interest in calculating blockage is upstream of the profiled frame members. For calculating the blockage effect, the drag, lift and other engineering values normally obtained from CFD are less important since they pertain to flow very close to the support members. However, flow separation in the wake of the profiled members must be captured since it may affect the velocity field farther upstream. Y+, an important value for valid results, should thus be kept within the range recommended in the software documentation. This may require fine meshing near the solid boundaries (cross member walls). The SST k-ω turbulence model was used with transitional turbulence (gamma theta model).

Initially, all simulations were considered two dimensional (2D) with a symmetry boundary condition for the vertical walls. Some 3D simulations showed that flows within an intake have a small lateral (left/right bank) component since the vertical walls are parallel. Calculations can thus be faster since the mesh is at least an order of magnitude smaller. Steady and, at a higher computational cost, unsteady simulations could thus be performed and results compared.

The simulation were done in a relative point of view, meaning that the blockage effect is calculated by changing only the distance between the walls (floor and ceiling) to simulate the blockage effect from a very long distance to a shorter one. The mesh was kept as constant as possible in the vicinity of the frame in order to minimize the error in the area of interest. A null blockage effect was simulated with an open boundary condition at the top and bottom of the simulation domain.

To reduce the mesh size related to the boundary layer near the walls, the opening at the emergency gate slot was not taken into account. A combination of symmetry and a constant velocity profile or a non-slip condition with a fully developed velocity profile was used respectively at the top/bottom and inlet boundaries.

Whenever possible, an initial simulation was run for a case where test data was available in order to gain confidence in the CFD results.

3. CASE 1 : FRAME SUPPORTING ASFM TRANSDUCERS

Case 1 is a test comparing measurements with the AS method to those with the acoustic transit time (ATT) method at EDF's Pinet power plant [13]. The discharge calculated for the AS method was 3,4% higher than for the ATT method, including a 0,32% reduction in discharge due to the blockage effect. The frame was made of two cylindrical pipes (Figure 2) with the two acoustic paths (broken yellow line) just upstream above and below the pipes. This frame was moved at different elevations in the measurement cross-section to sample the velocity profile.

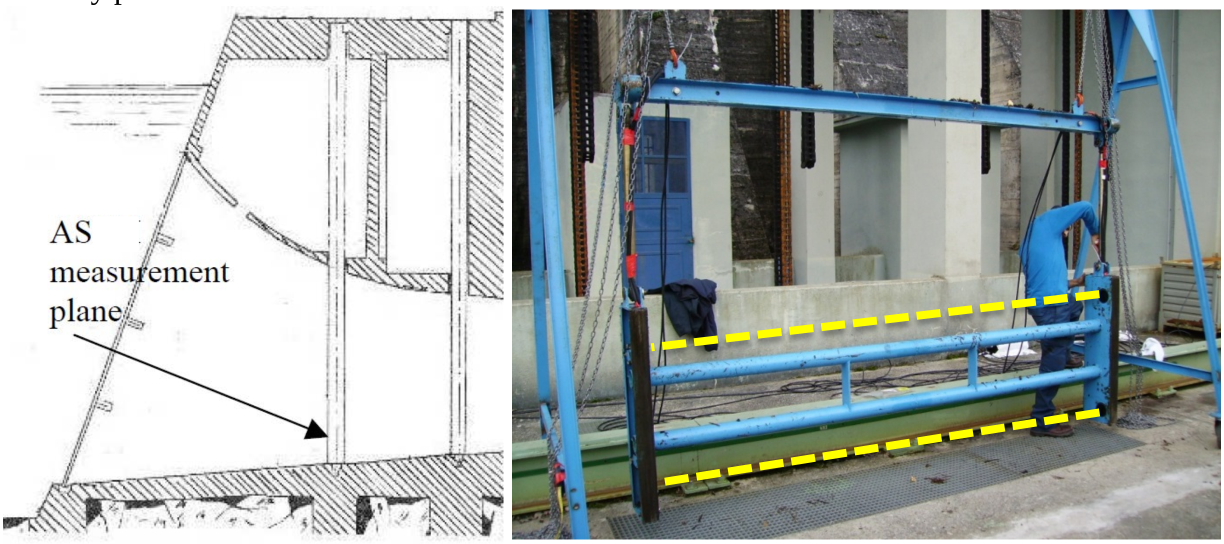

Figure 2: Intake profile at the measurement plane and AS transducers support frame showing the acoustic paths near the frame cross members

In the first simulation, flow around a single 2D cylinder was calculated and compared to the test data. The cylinder was 95 mm in diameter *D* with an inlet velocity *U* of 1 m/s for a Reynold number of $1{,}1 \times 10^5$. The first element thickness was 0,04 mm in order to perform an unsteady calculation to capture the Von Karman vortex street that normally appears under such conditions. Signs of the vortices can be seen in the results shown in Figure 3. The frequency *f* of vortex shedding can be estimated knowing the value of the Strouhal number *St* for that Reynold number:

$$St = fD/U = 0{,}22 \rightarrow f = 2{,}3 \text{ Hz}$$

The frequency calculated with the lift coefficient fluctuation obtained with CFD was 2,4 Hz.

The average drag coefficient from the CFD simulation was 1,35 (Figure 4) which is close to the test value of 1,4 for the same Reynolds number.

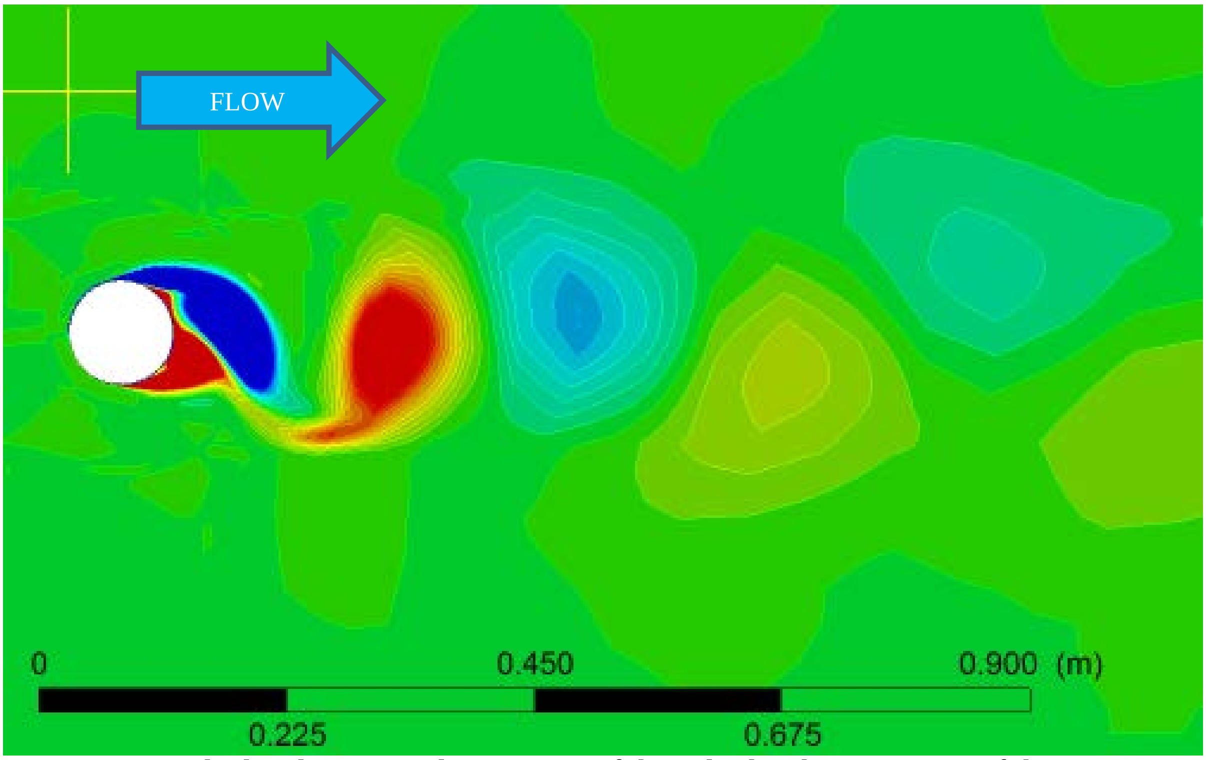

Figure 3: Calculated vorticity downstream of the cylinder showing signs of the vortices

As mentioned above, the region of interest is upstream of the cylinder where the transducers are installed (yellow cross in Figure 3). The difference between the local velocity and free stream velocity is the error *ErrU* due to the presence of the cylinder:

$$ErrU = \left(\frac{U(x,y)}{U} - 1\right) \times 100$$

CFD results give error of 3,9 % compared to a value of 4,5 % for a non-viscous theoretical calculation. Despite the lack of wake in non-viscous flow, the error is rather close meaning that the flow at the transducers' position primarily results from the geometry of the cylinder.

It is also useful to compare how steady and unsteady calculations differ. Both calculations were performed for a single cylinder with a blockage ratio of 4,75% (Figure 5). The error was 4,6% for the steady calculation compared to a mean of 4,5% for the unsteady calculation. Given this small difference, steady calculation results can be used with confidence despite unsteady flow. Note that the drag coefficient on the cylinder for the steady calculation was only 0,7, or about half that of the unsteady results (see Figure 4).

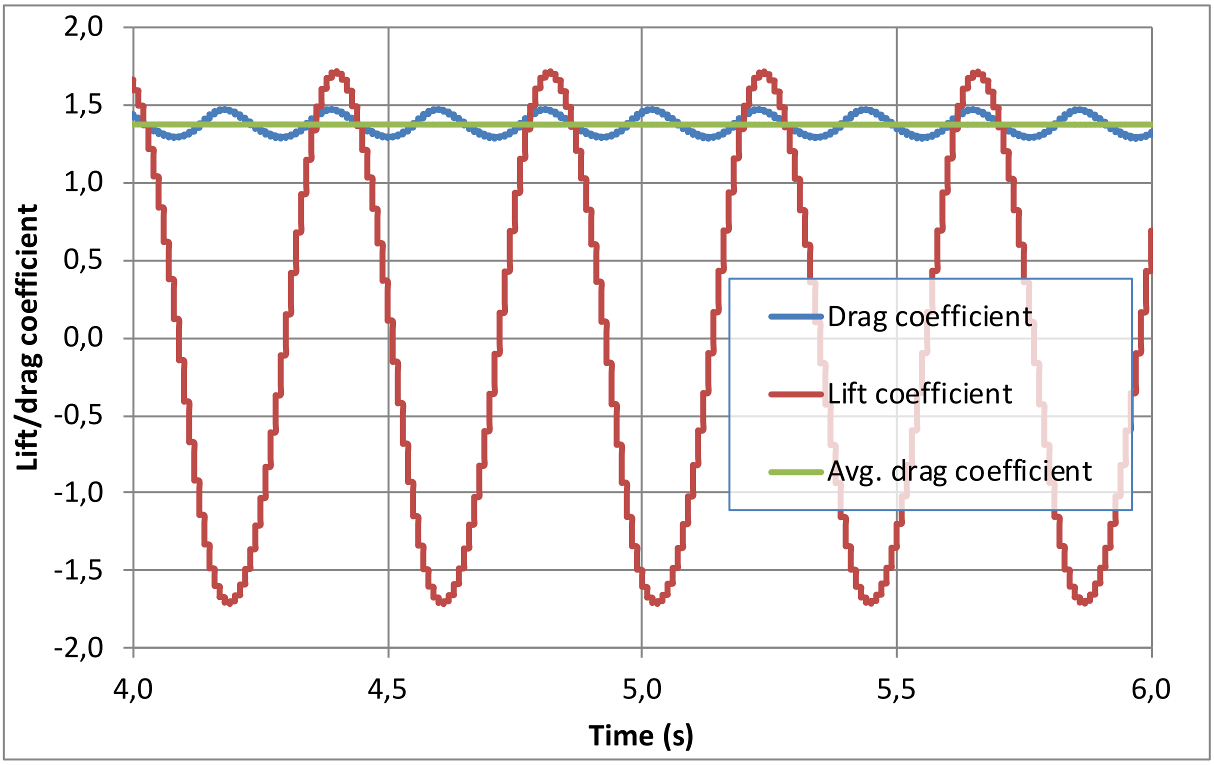

Figure 4: Single cylinder drag and lift coefficient vs time (s)

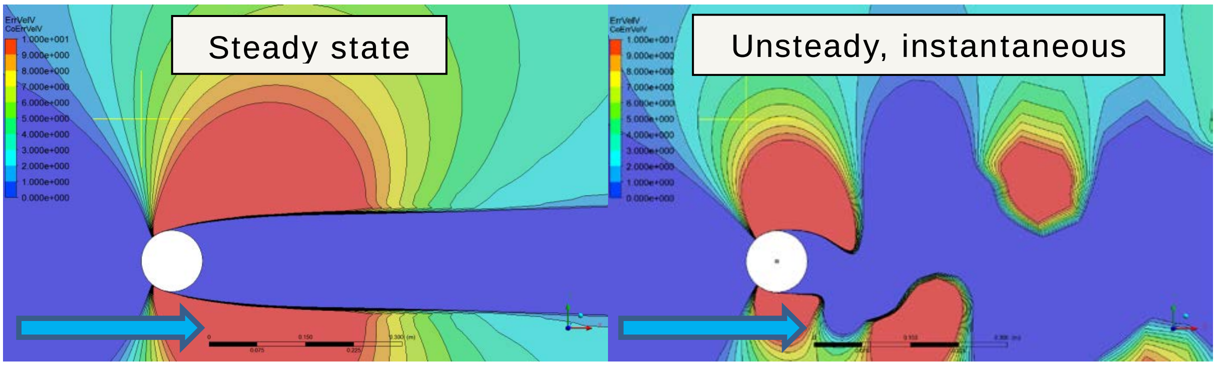

Figure 5: Comparison between steady and unsteady calculation

To calculate the real blockage effect of the AS transducer support frame in the Pinet power plant comparative test, a calculation domain containing two cylinders in the center of the cross-section with a blockage ratio of 1,3% was generated. As mentioned above, the domain was simplified to a straight section with 0° flow angle.

The results in Figure 6 clearly show that the AS transducers (the two rounded triangles) are in an area of accelerated (blue shows decelerated flow). At the AS transducers, the velocity error *ErrU* is 5 % (over-reading). Calculating at a second flow angle gives the same average value. Unlike current meters, which are mounted on a support for calibration and on-site measurements, AS transducers require no calibration so any error due to the support frame must be corrected during on-site measurements. There is clearly a local effect much larger than the known accelerated flow due to blockage by the frame.

Given the actual correction (0,32%) and new value (5%), the corrected AS discharge is 1,4% of the reference discharge. This value is within the uncertainty of the difference between those two discharges.

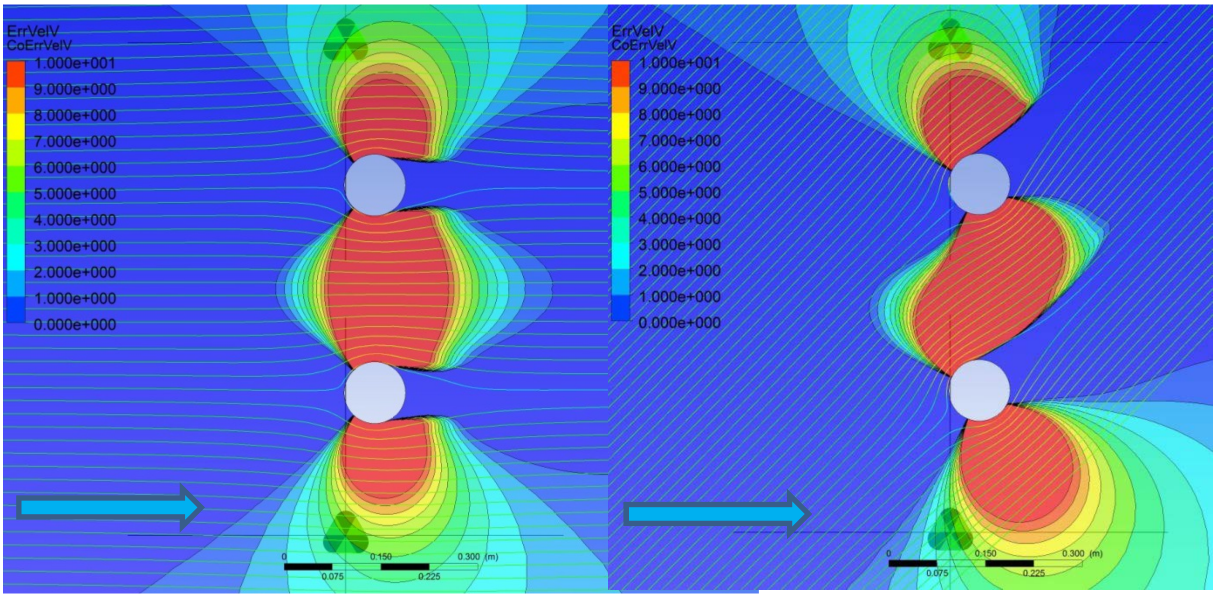

Figure 6: Calculation with two cylinders and a 1,3 % blockage effect at two flow angles

4. CASE 2: FRAME SUPPORTING CURRENT METERS

For current meter measurements at intakes, Hydro-Québec uses simple frames (Figure 1) made of two profiled bars (75 mm in chord by 35 mm in thickness) 1,3 m apart attached to two plates. The current meters are mounted on the lower bar with the propellers approximately 30 cm upstream of it. The frames are moved vertically with chain hoists to sample the entire velocity profile. The same profiled bar is used in calibrating the current meters. Two current meters are calibrated at once with the bar extending 0,7 m into the water of a calibration channel 5 m wide by 2,7 m deep.

The simulation domain comprises one or two profiled bars set at a 0° angle of attack. An open boundary was used to simulate free-stream flow. The flow velocity at the inlet of the domain varied between 1 and 6 m/s.

The relative velocity (local velocity divided by the free stream velocity) for a single bar in an unconstrained flow is shown in Figure 7. At the propeller location, the relative velocity is less than one. This represent the magnitude of the influence of the profiled bar on the current meter reading. This decelerated flow is partially taken into account in the calibration process.

Comparing CFD simulations with test data again gives confidence in the results. This is shown in Figure 8 where the drag coefficient for a profiled bar similar to that used by Hydro-Québec and tested by Chaix [14] is compare to CFD results for the two types of profiled bars. The drag coefficients are of the same order of magnitude and the CFD has the same rapidly decreasing value as a function of velocity.

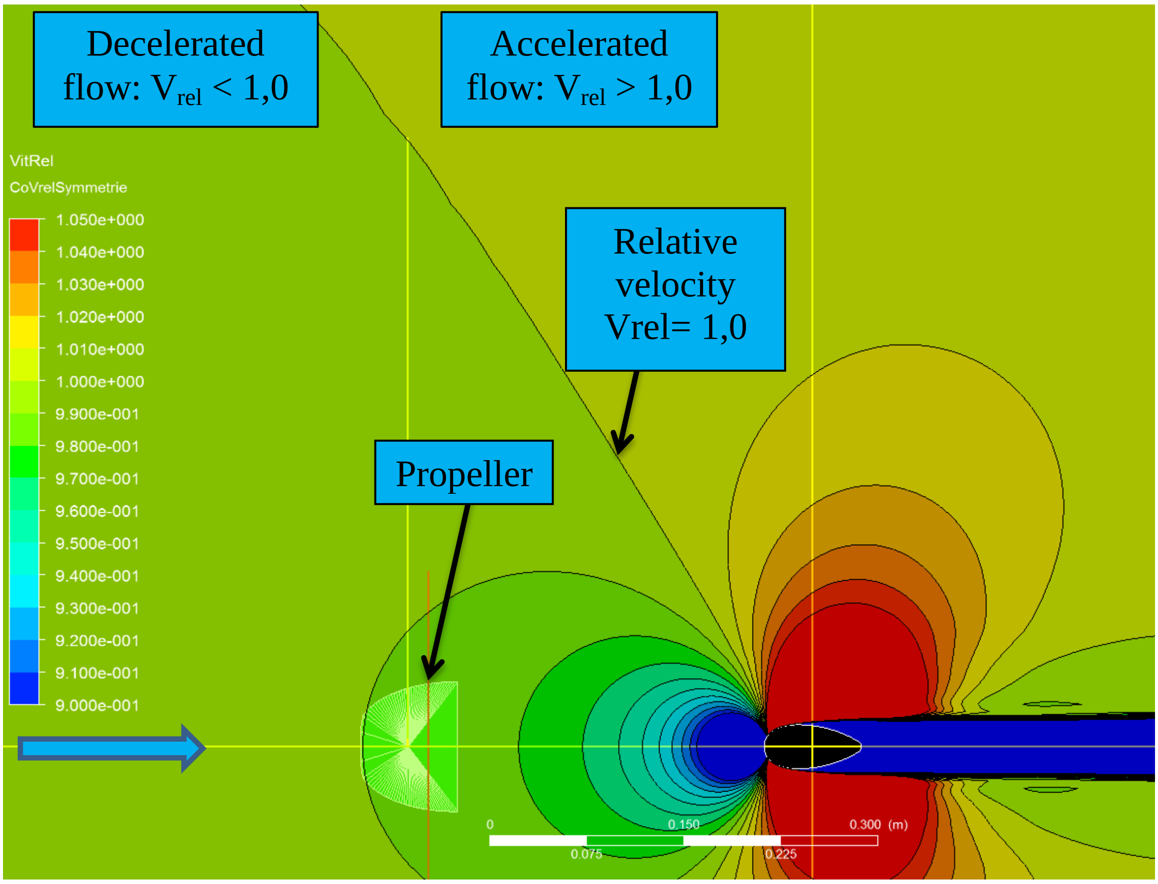

Figure 7: Relative velocity for a single profiled bar in a free stream flow

A first step in calculating the blockage effect for the Hydro-Québec frame is to calculate the change in relative velocity for one profiled bar in a constraint domain (top and bottom symmetry boundary condition) representing a set blockage ratio. Figure 9 shows the results of simulations at different velocities for steady and unsteady calculations for blockage ratios of 0% to 6%. The slope of the fitted curve represents the effect of the blockage ratio. Overall, the blockage effect is approximately 0,11 times the blockage ratio compared to 0,12 times in ISO 3354.

A final series of simulations was run to determine whether the position of the profiled bar in the measurement cross-section influences the blockage effect and to determine the effect of the second profiled bar on the one supporting the current meters (Figure 10). The results show that the position of the frame greatly influences the blockage effect, which is lower in the center of the measurement section and very close to the walls (0,15 m away). The effect near the walls gives an even lower velocity than for unconstrained flow with two profiled bars. This can be explained by looking at the results in Figure 7. A wall acts as like a mirror from the point of view of flow field meaning that it will be equivalent to the superposition of two flow fields, the first being a normal flow field without wall and the second one being the same but reflected at the position of the wall. Thus, the vertical line from the propeller to the line where the relative velocity is 1,0 (about 47 cm above the propeller) is located in an area of decelerated flow. A wall placed within half this 47 cm distance will thus reduce the current meter velocity reading.

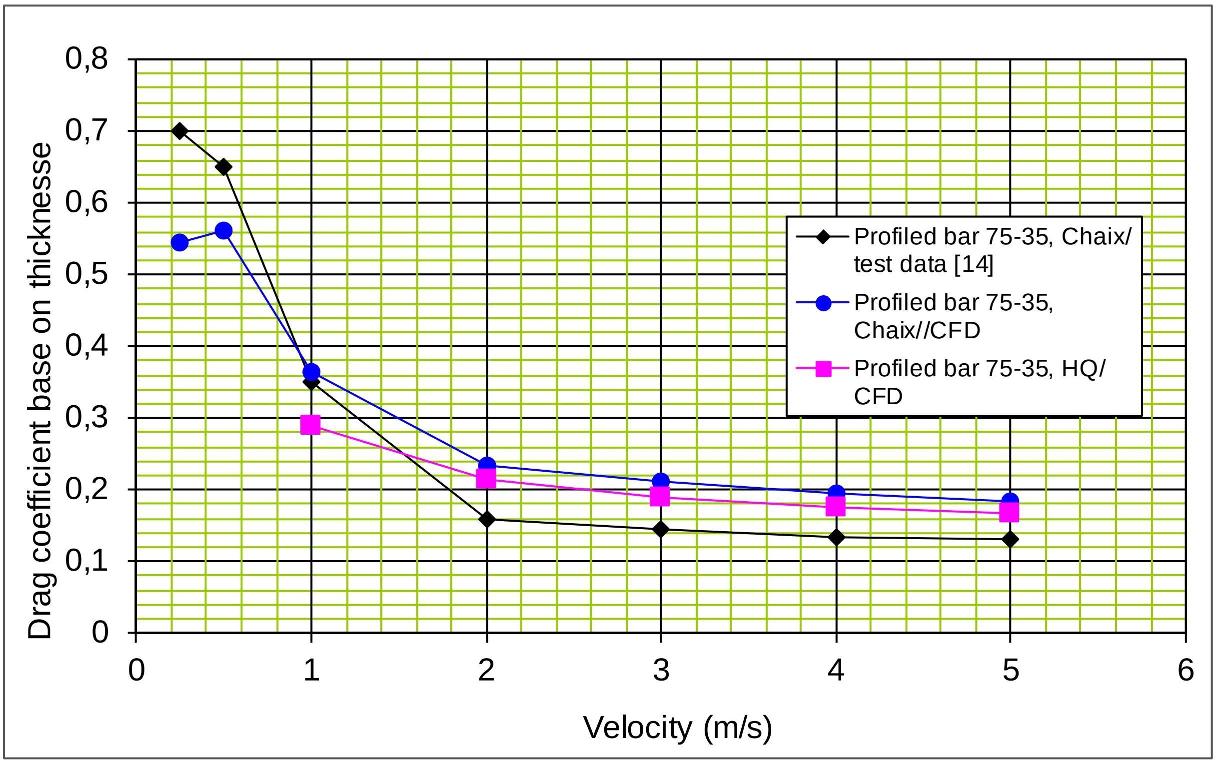

Figure 8: Drag coefficient based on thickness for different profiled bars

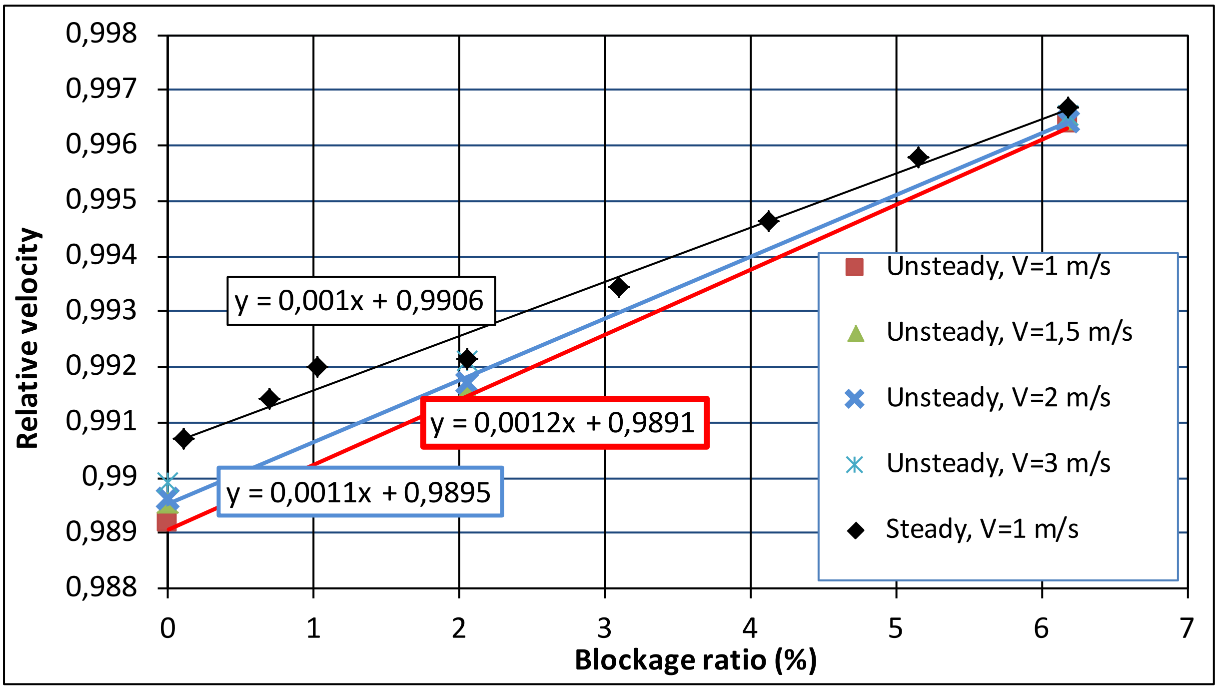

Figure 9: Blockage effect for one profiled bar at different inlet velocities

The relative velocity peaks near 45 cm from the wall (possibly not the highest value) and decreases towards the middle of the measurement section. This can be explained by a mirror effect of the walls (top or bottom) that decreases as the profiled bars approach the middle of the cross-section. Detailed examination of the CFD results shows that the relative velocity asymptotically decreases toward 1,0 as the distance from the propeller increases.

In this study, the opening normally present at the top of the measurement cross-section was ignored. It can be concluded from the results that the blockage effect at the top and bottom will differ since the mirror effect will be partial at the top, thus modifying the relative velocity. In addition, when the upper bar is outside the flow, its flow field disruption will be close to zero.

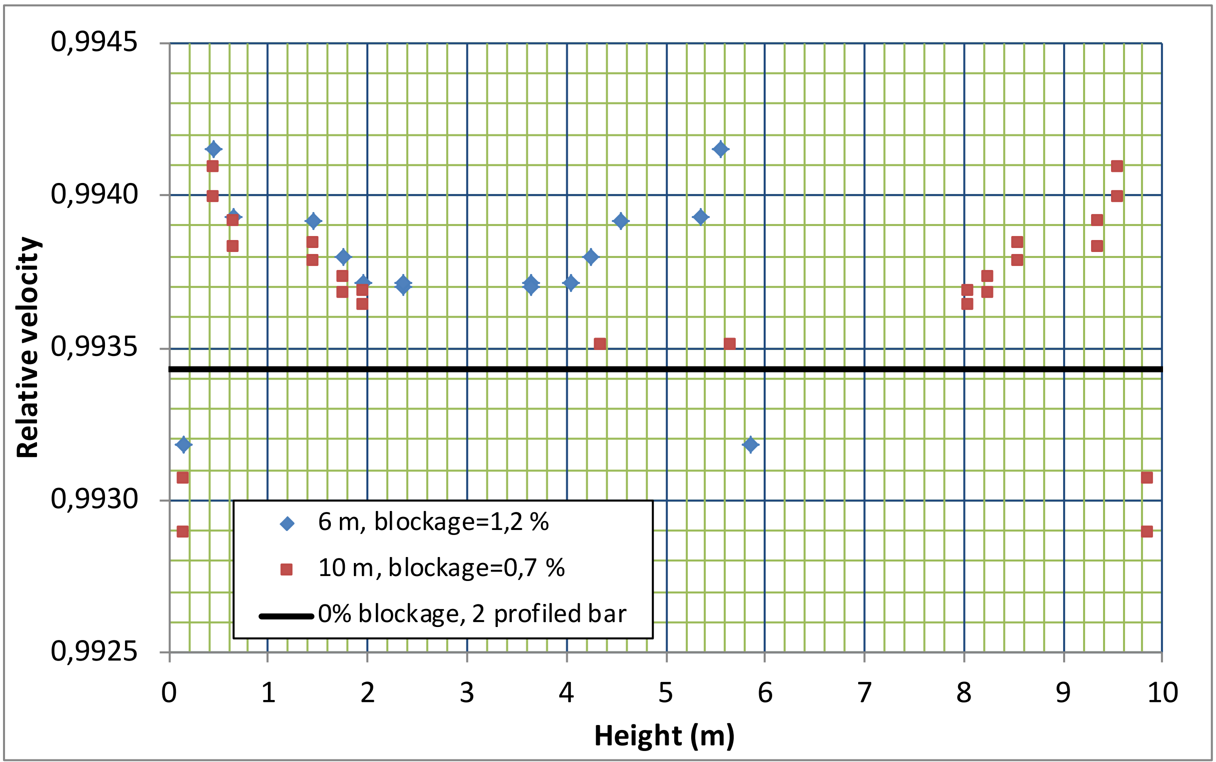

Figure 10: Relative velocity vs. position of the current meter support frame for two blockage ratio

5. FUTUR SIMULATIONS

All CFD simulations in this paper were run with some simplification of the real flow. Several important issues must be addressed for a more thorough analysis.

All simulations were run with the flow in line with the profiled bar. Most low-head power plant intakes have a converging flow with a measurement cross-section in which the flow angle will vary with position. It is thus important to study the influence of the angle of attack of the flow relative to the profiled bars since the frontal area may be much larger and affect the flow angle itself.

As mentioned earlier, the opening at the top of the measurement cross-section was ignored in the meshing domain. The blockage effect will differ due to inexistence of the wall there and this will change the overall value of the effect.

Another important factor to evaluate is the blockage effect during current meter calibration. CFD tools are capable of also calculating this blockage effect. For this, a 3D domain with a free-surface two-phase flow domain must be considered.

.

Because calculating every situation can be time consuming, an interesting approach is to develop simplified rules from an unconstrained flow field calculation that can be used with a method of flow superposition as was done in [14]. Another approach would be to fully automatize mesh generation, solving and post processing.

The blockage effect discussed in this paper was calculated only for the main structural members of the support frame, ignoring the current meters and other parts of the frame (cables, center bar, etc.). The blockage effect of the current meters, normally less than that of the frame, can also be calculated with CFD.

Lastly, an important task will be to estimate the uncertainty of CFD results in order to lower the uncertainty related to the correction of the discharge with this method. The actual uncertainty related to the correction of the blockage effect is relatively large (66 % of the correction), which is a reason for limiting the blockage ratio to 6 %. In hydraulic turbine performance calculations, CFD uncertainty is commonly about 5% and never exceeds 10%. There is no reason for higher uncertainty in calculating the blockage effect. Lower uncertainty would make it possible to perform measurements with a blockage ratio higher than 6%, advantageous for testing at small power plants.

6. CONCLUSION

This paper showed how CFD can be used to calculate the real blockage effect when making discharge measurements at a power plant intake. CFD has many advantages over the ISO 3354 formula since the actual conditions can be calculated. Results show that high confidence can be placed in the CFD method. More simulations must be run to compute the real value for discharge correction and CFD is a great tool to perform that task.

References

[1] G. Proulx and E. Cloutier, “Hydro-Québec Experience with Discharge Measurement in Short Converging Intake”, HydroVision 2011, Sacramento, CA, USA.
[2] P. Lamy and J. Néron, “A Different Approach in Measuring Individual Turbine Efficiencies in Multiple Unit Power Plants”, Water Power XIII, Buffalo, NY, 2003.
[3] J.W. Taylor, G. Proulx, J. Lampa, Turbine flow measurement in intakes – a cost-effective alternative to measurement in penstocks, Hydro 2011,
[4] J. Lampa, D.D. Lemon, P. Lamy, J.W. Taylor, Turbine flow measurement for low-head plants – Owners’ options for the 21st century, Hydro 2007, Granada, Spain
[5] R.I. Munro, J. Walsh, Kootenay canal flow comparison test using intake methods – background and goals, Hydro 2010, Lisbon, Portugal
[6] C.W. Almquist, J.W. Taylor, J.T. Walsh, Kootenay Canal Flow Rate Measurement Comparison Test Using intake methods, HydroVision 2011, Sacramento, CA, USA
[7] J.W. Taylor, C.W. Almquist, J.T. Walsh, Results of Kootenay canal flow comparison test using intake methods, Hydro 2010, Lisbon, Portugal
[8] G. Proulx, Kootenay Canal Plant Comparative Flow Measurements: Current Meter method, Hydro 2010, Lisbon, Portugal, 2010
[9] D.D. Lemon, J. Lampa, Cost-effective turbine flow measurement in short intakes with acoustic scintillation, Hydro 2004, Porto, Portugal
[10] G. Proulx and al., The Acoustic Scintillation Method: Two Cases Studies Of Comparative Measurements, IGHEM 2014, Itajubá, Brasil

[11] ISO-3354, Measurement Of Clean Water Flow In Closed Conduits – Velocity-Area Method Using Current Meters In Fill Conduits And Under Regular Flow Conditions
[12] IEC 60041, Field Acceptance Tests to Determine the Hydraulic Performance of Hydraulic Turbines, Storage Pumps and Pump-Turbine
[13] B. Reeb and al., Case Studies of Discharge Measurement using the Acoustic scintillation Flow Metering Technique, Hydro 2007, Granada, Spain
[14] B. Chaix, Effets d'interférence entre moulinets hydrométriques, support et parois lors de mesures de débits, École Polytechnique Fédérale de Zurich, Zurich, 1972