Computational Fluid Dynamics (CFD) Modeling of Run-of-River Intakes

Jose Vasquez, Kara Hurtig, Brian Hughes
Northwest Hydraulic Consultants Ltd., North Vancouver, Canada.

ABSTRACT

This paper presents four case studies of run-of-river intakes where computational fluid dynamics (CFD) modeling was applied to simulate their three-dimensional (3D) flow hydraulics in order to assess different aspects of their designs. The first case study looked at potential construction cost savings by reducing excavation downstream from the powerhouse and spillway and its potential impact on flow hydraulics. The second case analyzed different intake layouts to minimize headlosses and provide more uniform velocity distribution in front of the intake. The remaining case studies assessed the potential for sediment ingestion into the intake, either by looking at the computed 3D flow paths or by tracking the trajectories of simulated sediment particles. The results show that CFD analyses can provide good descriptions of flow hydraulics and reasonable qualitative information on sediment movement, making CFD a valuable tool for assessing alternative designs, especially at the feasibility level.

INTRODUCTION

Over the last few years, environmental concerns regarding the impact of large dams and reservoirs on natural rivers has led many hydroelectric projects to use run-of-river intakes as an alternative to intakes in deep reservoirs. Since run-of-river intakes lack the upstream ponding effects of large reservoirs, approach flow velocities tend to be higher and flow conditions more turbulent, which in turn increases the potential for sediment to be ingested at the intake (Bouvard 1992). The complex three-dimensional (3D) hydraulics and the possibility of sediment entering the intake usually demand physical model studies to ensure a proper design. However, recent developments in Computational Fluid Dynamics (CFD) are allowing some of the hydraulic aspects of run-of-river intakes to be evaluated by CFD models as a complement – or in some cases an alternative - to physical modeling.

CFD analysis is a numerical approach by which the governing equations that describe fluid flow are solved at thousands (often millions) of discrete points within the computational domain; hence providing hydraulic engineers and designers with the ability to evaluate the performance of different designs (Hughes et al. 2012).

This paper attempts to illustrate by the use of test cases, the possible applications of CFD modeling for assessing and designing hydroelectric run-of-river intakes and associated structures. Due to the confidential nature of the developments, project and client names have not been included in this paper.

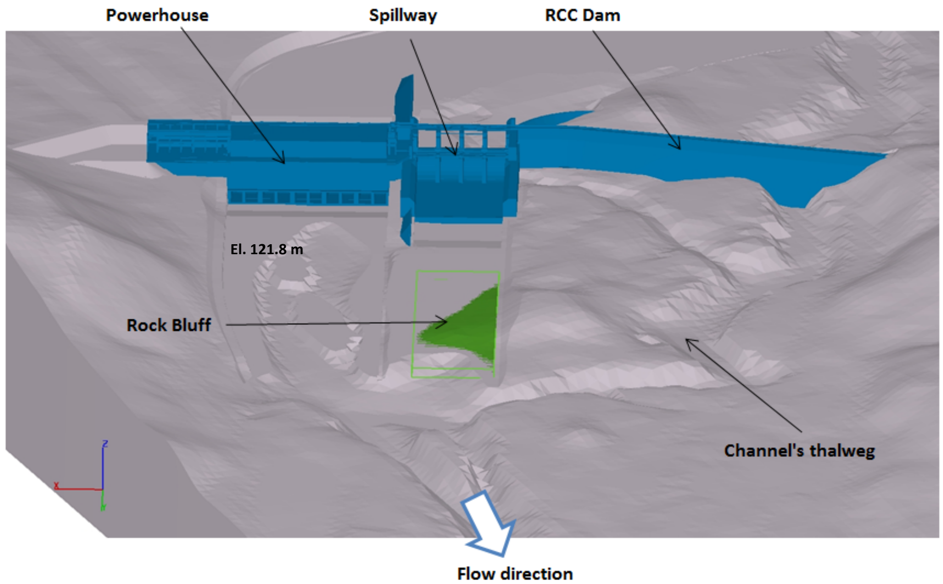
CASE STUDY 1 - TAILRACE CHANNEL MODIFICATIONS

This case study involves the application of CFD modeling to assess potential economic savings for a proposed design-build-operate hydropower project. The tailrace of the dam's spillway and powerhouse is to be excavated on rock on the right side of the river channel. One of the companies bidding for the project was interested in knowing whether construction costs could be lowered by reducing rock excavation in the tailrace channel without adversely affecting the performance of the spillway or powerhouse. Options that were being considered included not removing a rock bluff located within the tailrace channel (Figure 1.a), raising the tailrace invert elevation by 5.8 m (Figure 1.b), or both. Being able to incorporate any of these options would lead to significant cost savings for the project. However, they may produce undesirable effects for the hydraulic performance of the powerhouse or spillway stilling basin. Since this analysis was intended for a pre-bidding assessment, physical modeling was not warranted. Also, two-dimensional depth-averaged modeling was not considered suitable because of the 3D flow patterns in the stilling basin and downstream from the powerhouse draft tubes. Therefore, the analysis was performed using CFD modeling in order to:

- Assess whether leaving the rock bluff in place produces undesirable flow conditions that could negatively impact the performance of the generating units (for example, excessive flow oscillations, backwater impacts, return currents, etc.).
- Evaluate the characteristics of the hydraulic jump; for example, whether it is contained within the spillway stilling basin or swept downstream.
- Assess the energy dissipation in the stilling basin and the exit flow velocity (as a metric for downstream scour potential).
- Assess potential interference between the powerhouse and spillway when operating simultaneously.
- Assess the effects of raising the tailrace invert elevation on the powerhouse running at maximum capacity.

The CFD results showed that leaving the rock bluff in place (Figure 1.a) had no discernible effect on the hydraulic performance of the spillway or powerhouse. The modified tailrace channel downstream, with an invert elevation increased by up to 5.8 m (Figure 1.b), caused only a modest increase of 5 cm in the water levels immediately downstream of the powerhouse and no other noticeable effects. The conclusion of the CFD analysis was that the two options to reduce rock excavation, and hence construction costs, could be carried out without detrimental effects on the hydraulic performance of the structure, thus reducing construction costs.

a) Rock bluff left in place



b) Tailrace channel excavation reduced by 5 m

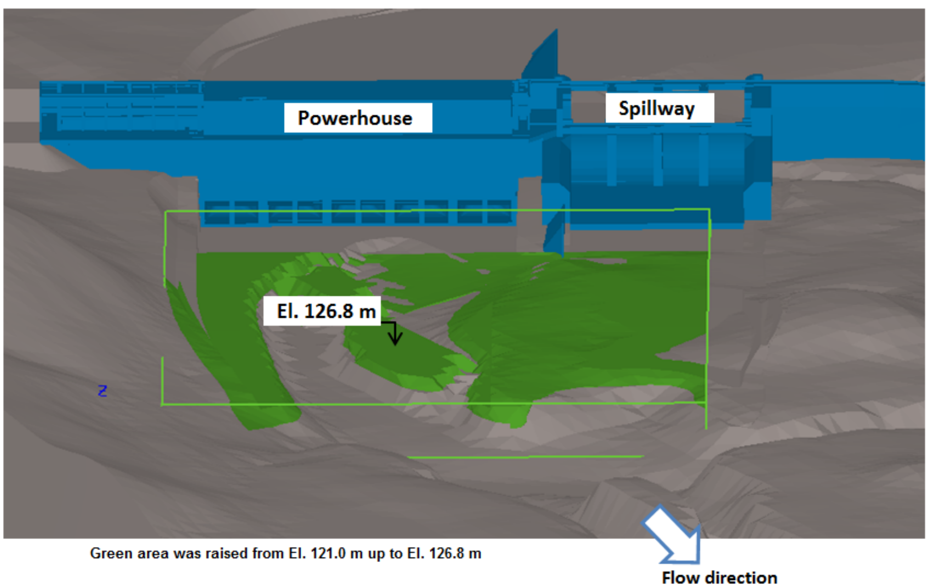


Figure 1. Tailrace channel with (a) rock bluff left in place and (b) tailrace channel excavation reduced by 5 m.

CASE STUDY 2 - POWERHOUSE INTAKE LAYOUT

This case study involves a new powerhouse proposed for construction at an existing lock and dam (Figure 2). The new powerhouse will require excavation of an approach channel to connect the existing reservoir with the proposed intake. CFD modeling was used to study alternative powerhouse approach channel layouts and recommend a design for further project development and testing in a physical model. CFD testing included preliminary design development to optimize hydraulic conditions such as head losses and flow uniformity at the powerhouse intake. Detailed evaluation of three proposed powerhouse approach channel layouts at various flows was completed. Velocity data at the face of the intake, approach flow patterns, and headloss through the channel for three approach channel alternatives were evaluated and compared. The alternative that provided more uniform flow patterns approaching the intake and less headloss through the channel, while maintaining reasonable construction costs, was selected and recommend for testing in the physical model.

The initial CFD testing showed that flow conditions in front of the intake were highly non-uniform, with a large eddy present in the approach channel (Figure 2), which also caused increased energy losses. It was recognized that the main reason for the uneven flow distribution was the presence of an existing spur dyke located on the left bank upstream from the intake (Figure 2). Additional factors were the alignment of the approach channel, the location of the intake and the invert elevation. Several design modifications were tested to improve the flow conditions. The optimized layout had the face of the intake moved farther upstream (closer to the reservoir), curved guiding walls, the spur dyke removed, and a lowered invert elevation along the right side of the approach channel entrance (Figure 3). This optimized geometry eliminated the large eddy in front of the intake, leading to a more uniform approach flow (Figure 3) and minimum energy losses

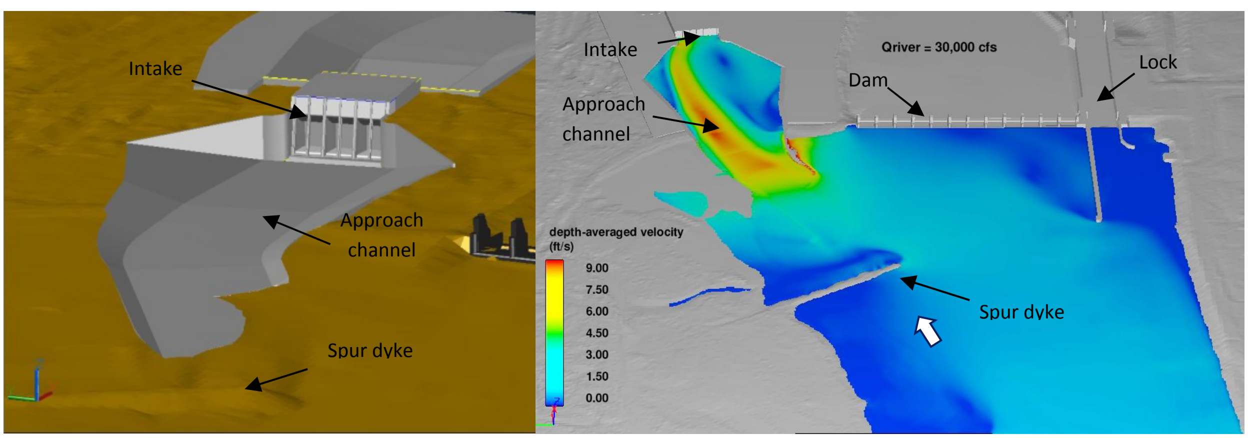


Figure 2. Initial design layout of powerhouse intake. Notice spur dyke upstream of intake causing highly non-uniform approach flow conditions.

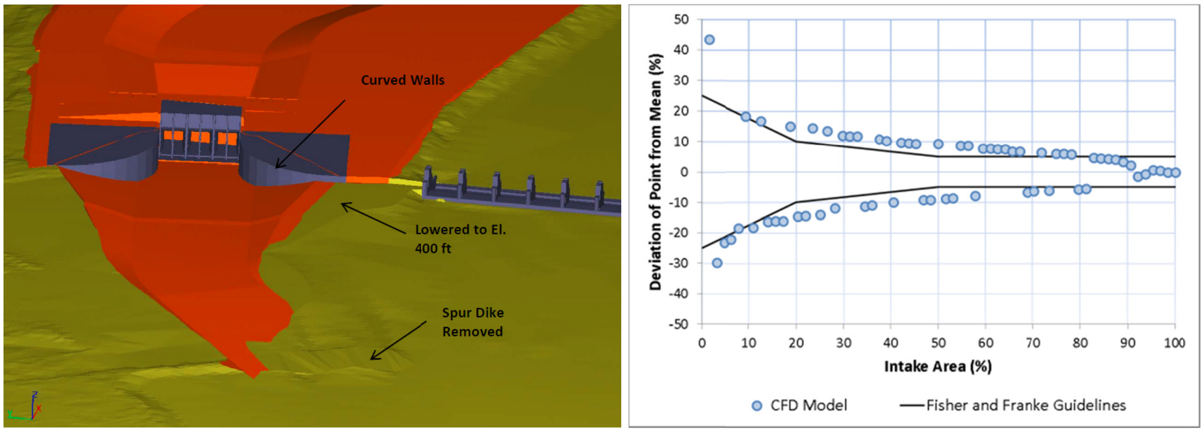


Figure 3. Optimized intake geometry and the velocity distribution in front of the optimized intake computed following Fisher and Franke (1987) guidelines.

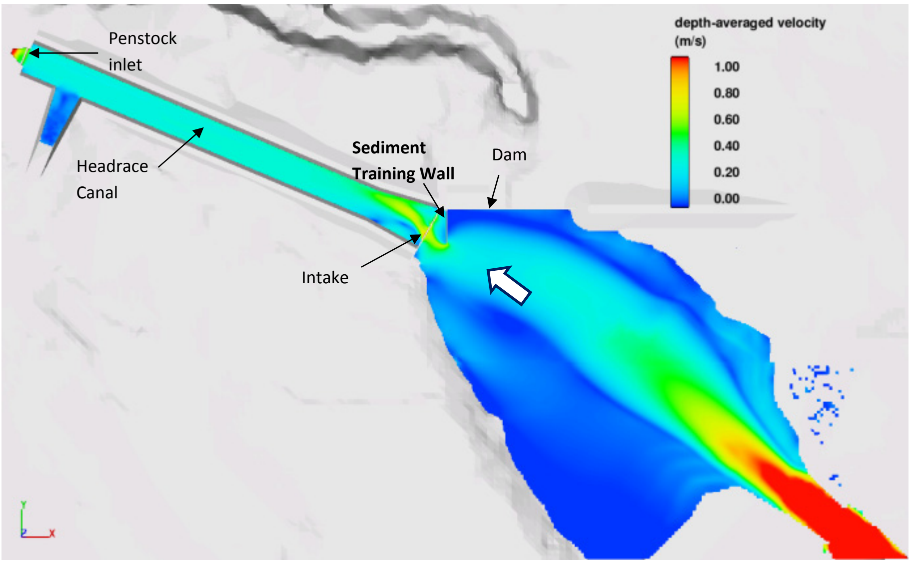
CASE STUDIES 3 AND 4 - SEDIMENT TRANSPORT IN INTAKES

One of the main sediment-related problems in run-of-river intakes is the possibility of sediment ingestion by the intake. Sediment transport processes in run-of-river intakes are still too complex for a complete and satisfactory description using CFD models alone, and physical models are still required for a detailed evaluation of the sediment transport processes. However, CFD models can provide valuable qualitative information when comparing different intake designs. For example, it is known that sediment transport and re-suspension is more likely to occur if bed shear stress and turbulence intensity are higher. Therefore, a CFD model could be used to compare the flow patterns of alternative designs and provide some insight about sediment movement. Another possibility is to use CFD particle tracking to simulate the movement of sediment (Vasquez 2010). Two case studies using both approaches are described below.

Case Study 3 - Hydraulics of sediment training wall

In order to remove sediment deposits in front of the intake, a sluiceway gate is normally located crossing the dam axis and adjacent to the intake with the purpose of flushing sediment deposits. To create a flushing channel and increase the flushing efficiency, it is customary to add a training wall that separates the flushing channel from the rest of the dam (Bouvard 1992). Under typical flow conditions with the dam aligned perpendicular to the river flow direction, the training wall is parallel to the approach flow direction and has the positive effect of increasing flushing efficiency. However, in the test case shown in Figure 4, the approach flow was almost parallel to the dam, attacking the training wall with an acute angle. The CFD analysis showed that because of this skewed approach flow direction, the training wall created very turbulent flow conditions with increased local velocities. The 3D streamlines showed that when the training wall was present, some streamlines dived down to the bottom and then rose again towards the surface, before entering the intake (Figure 5). Such a flow path would make it possible for the flow to pick up sediment from the bottom, re-suspend it and then transport it into the intake. Based on these results, it was decided not to include a full-height training wall.

a) With training wall



b) Without training wall

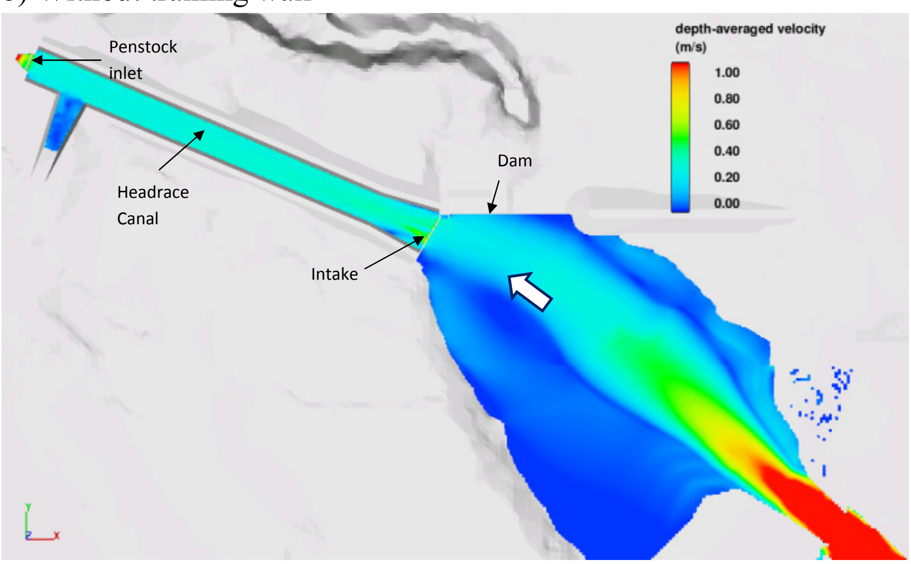
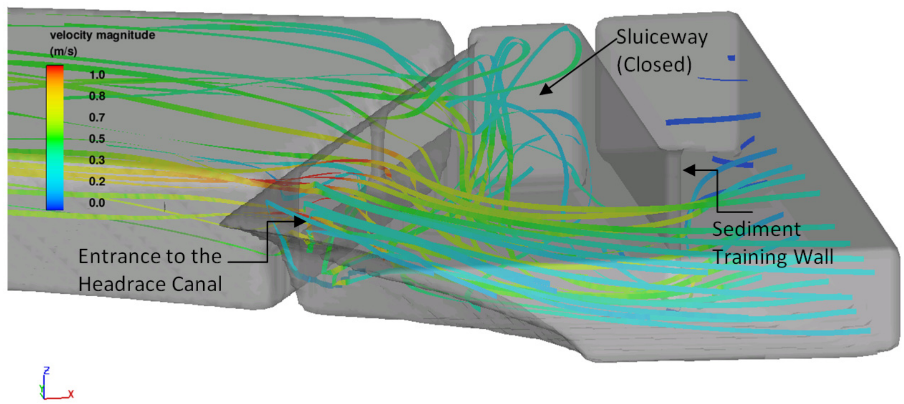


Figure 4. Plan views of depth-averaged flow velocity (a) with training wall and (b) without training wall.

a) With training wall



b) Without training wall

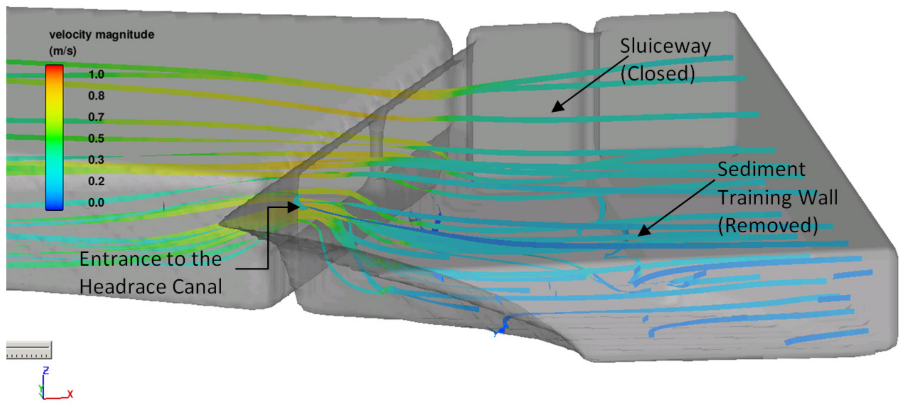


Figure 5. 3D streamlines through the intake (a) with training wall and (b) without training wall.

Case Study 4: Sediment particle tracking

Another possibility to investigate the movement of sediment past run-of-river intakes is the use of Lagrangian particle tracking (Vasquez 2010). In the CFD model the particles are assumed as spheres with size and density similar to that of the natural sediment. The particles are released upstream and their paths tracked, making it possible to qualitatively assess the proportion of sediment entering into an intake and compare the performance of alternative designs. To illustrate this potential application, the final case study involves a large run-or-river intake located in a steep mountainous river, where coarse sediment (up 70 mm) has been observed to be transported in suspension during flood events. Figure 6 shows the general layout of the structure and a screenshot of CFD particles between 1 and 10 mm. Because of curvature effects and secondary currents, a large portion of the incoming particles are shown to be diverted into the intake. Several design modifications were tested using CFD modeling and eventually physical modeling; the final solution included a desanding facility in the forebay area.

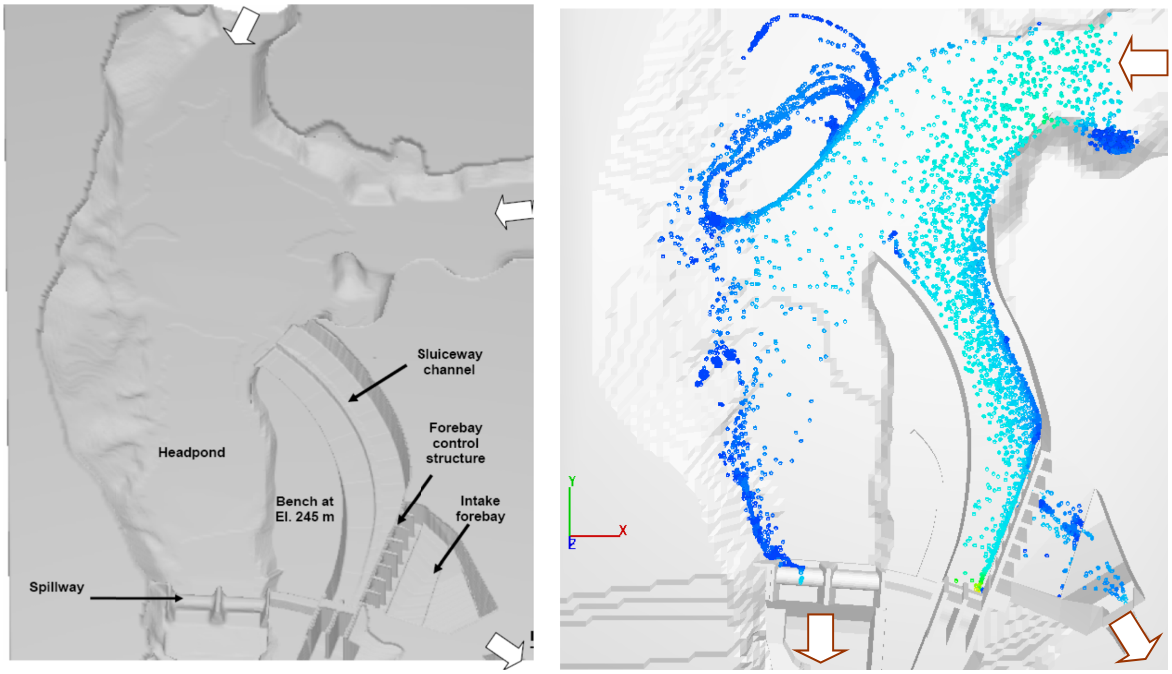


Figure 6. Run-of-river layout and computed particles tracks

SUMMARY AND CONCLUSIONS

Run-of-river intakes are usually subject to more challenging flow conditions compared with intakes located in large reservoirs and are usually subject to higher approach flow velocities and sediment loads. Since flow is highly three dimensional at the intake and in the vicinity of the dam and stilling basin, these structures typically require the use of physical models to verify their performance. However, CFD analysis is increasingly becoming a complementary tool for the hydraulic design of run-of-river intakes under various flow conditions. CFD analysis is particularly useful during early stages of a hydropower project, even during the bidding phase, when physical modeling may not be warranted.

Provided that a computational mesh with adequate resolution is used, CFD models can provide reasonable predictions of the flow patterns to be expected in most hydraulic structures. Although sediment transport processes cannot be simulated by CFD models with the same level of confidence as provided by physical models, CFD models can provide some qualitative insight about the potential movement of sediment that can be useful when comparing the performance of different alternatives.

REFERENCES

Bouvard, M. (1992). "Mobile barrages and intakes on sediment transporting rivers". IAHR Monograph. Balkema, Rotterdam.

Fisher, R. and Franke, G. (1987). "The Impact of Inlet Flow Characteristics on Low Head Hydro Projects". Conference Proceedings of Water Power '87. pp. 1673-1680.

Hughes, B.H., Vasquez, J.A. and Hurtig, K.H. (2012). "Application of Computational Fluid Dynamics Analysis for Hydroelectric Projects".HydroVision Brasil 2012.

Vasquez, J.A. (2010). "Assessing sediment movement by CFD sediment particle tracking". 9th Federal Interagency Sedimentation Conference, FISC 2010, Las Vegas, NV. June 27-July 1, 2010.

AUTHORS

Jose (Pepe) Vasquez is an Associate at Northwest Hydraulic Consultants (NHC) in Vancouver, British Columbia Canada. He is a specialist in sedimentation and advanced numerical modeling, including 1D, 2D and 3D methods. Dr. Vasquez was the lead CFD modeler for the majority of the applications described herein and can be reached at JVasquez@nhcweb.com.

Brian Hughes is a Principal at Northwest Hydraulic Consultants (NHC) in Vancouver, British Columbia, Canada. He is the Director of NHC's hydraulic modeling and served as the Project Manager and Principal in Charge for several of the applications described in this paper. Brian can be reached at BHughes@nhcweb.com.

Kara Hurtig is a Senior Engineer at Northwest Hydraulic Consultants (NHC) in Vancouver, British Columbia Canada. She is a physical and numerical modeler who conducted the 3-dimensional CFD analyses for several of the projects described in this paper. Kara can be reached at KHurtig@nhcweb.com.