Power Intake Velocity Modeling Using Flow 3D at Kelsey Generating Station

By Shuang Ming Wang, Ph.D, P.Eng., Manitoba Hydro, Winnipeg, Manitoba, Canada
Kevin Sydor, MSc., P.Eng., Manitoba Hydro, Winnipeg, Manitoba, Canada

Manitoba Hydro has initiated a Supply Efficiency Improvement Program to update its existing generating stations. Under this program, potential upgrade projects are evaluated that would allow the utility to increase energy production by reducing energy losses within existing powerhouses and intake works. As a part of this program, intake conditions associated with Manitoba Hydro's 224 MW Kelsey Generating Station were recently studied to assess the performance of the intake both prior to, and following the re-runnering of the project's seven units.

The Kelsey Generating Station is located on the Upper Nelson River between Sipiwesk and Split Lake, as shown in Figure 1. The plant consists of 7 units with a total capacity of 224 MW. The plant discharge is currently approximately 1700 m^3/s and after re-runnering the units, is expected to be increased to 2200 m^3/s. Approach conditions to the intake are not ideal, and even under existing conditions the presence of a rock knob in the approach channel, located about 250 m upstream on the east side of unit 7, creates a large, turbulent eddy. The efficiency of units 6 and 7 is affected by the non-uniform flow in the intake channel, and the re-runnering of the turbine will need to consider these flow characteristics.

Ideally, the intake and channel must be designed to facilitate water flow to the turbine, minimize local head loss and maximize the hydraulic efficiency of the equipment. The expected intake velocity distribution is very important, as it will allow turbine manufacturers to optimize the design of the machinery.

Engineers at Manitoba Hydro used a comprehensive three dimensional numerical model to evaluate the intake velocity distribution for the Kelsey Generating Station, both under existing and post re-runnering conditions. The simulation results were then provided to the turbine manufacturer for their use in selecting and optimizing the turbine equipment.

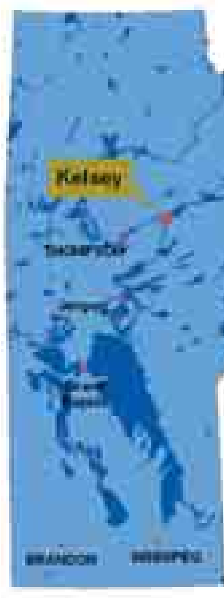


Figure 1: Kelsey Generation Station

Flow 3D

Numerical modeling has developed into a powerful tool that can be used by engineers to quickly and inexpensively explore different design options as well as demonstrate how a water resource project can be constructed and/or operated more efficiently. Manitoba Hydro has used CFD analysis to advance the hydraulic design of many projects for several years. Several papers have been published and present a number of specific design examples (Teklemariam, et al. 2000 & 2002, Groeneveld, et al. 2005, St. Laurent, et al. 2005).

The FLOW-3D software was selected for used in this evaluation. The Flow 3D program, developed by Flow Science Incorporated of Sante Fe, New Mexico, USA, is a CFD model capable of simulating the dynamic and steady state behavior of liquids and gases in one, two or three dimension. It does so through solution of the complete Navier Stokes equations of fluid dynamics. It is applicable to almost any type of flow process and capable of simulating free surface flow. The program utilizes specialized algorithms to track the location of the water surface over large and small spatial and temporal variations. These capabilities make the model well suited for simulating varied and complex flow conditions, which typically occur in a variety of hydraulic design and analysis problems.

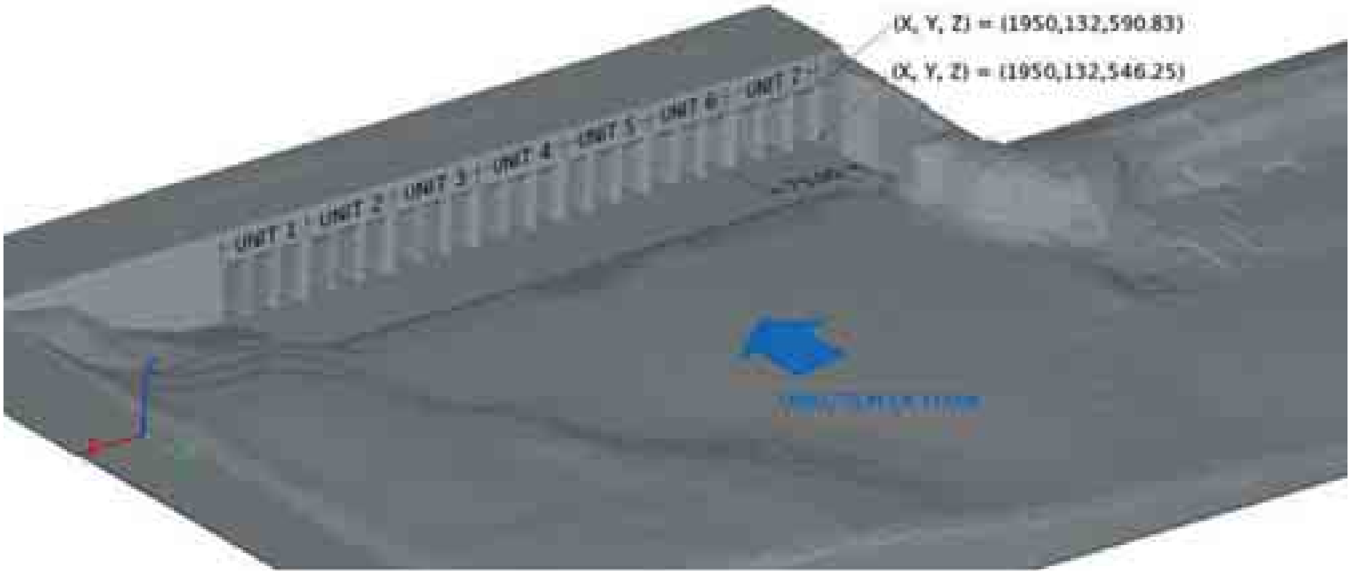


Figure 2: Forebay Bathometry and Intake Layout

Flow 3D has been used extensively by Manitoba Hydro for complex flow simulations. In this study, it was used to simulate the powerhouse intake flow conditions.

Model Set Up

The model was setup based on existing drawings in the Manitoba Hydro drawing management system. All major intake components were modeled, but smaller features like the stop log/intake gate guides and trash racks, were not included in the model. Figure 2 below shows the forebay bathymetry and the powerhouse intakes. There are a total of seven units, each with three openings, to direct the flow to the turbine. The three openings for each unit are labeled from east to west as A, B and C. Three inter-block meshes were used in the simulation to optimize overall run times. The numerical mesh for areas in the upstream approach channel were given a relatively coarse discretization, whereas a much finer mesh was setup within the intake to provide more detailed velocity information for the turbine manufacturer. A constant water level was applied as the upstream boundary condition, and a mass sink was used to simulate the outflow through the power house.

Model Test

A few test runs were initially conducted to determine the proper way to simulate the outflow from the powerhouse. Due to the effect of the rock knob, the flow is quite different through each opening, particularly for units 6 and 7. In order to properly model the inflow to the powerhouse scroll case, a separate mass sink was applied to each water passage opening. The flow through each unit was divided such that openings A, B, and C received 40%, 36% and 24% of the total flow respectively. These flow ratios were determined based on the average of the measured results for units 1 to 5 under full gate operation, during turbine performance testing in 1990. Although units 6 and 7 were only tested up to 40%, the same ratio was applied to all seven units.

A velocity distribution for each opening was available for the pre-rerunnering condition at full gate from the 1990's tests. Visual analysis of the model results also shows a close reproduction of field conditions. In particular, the model was able to simulate the formation of a vortex in front of units 6 and 7 caused by the "rock knob" just upstream of these units. This is similar to intake flow fields observed during the metering

Table 2: Outflow (m^3/s) for each opening for Cases 1-3

Cases	Opening	Unit 1	Unit 2	Unit 3	Unit 4	Unit 5	Unit 6	Unit 7
Units 1-7 RR at FG	A	132.2	132.2	132.2	132.2	132.2	132.2	132.2
	B	119.0	119.0	119.0	119.0	119.0	119.0	119.0
	C	79.3	79.3	79.3	79.3	79.3	79.3	79.3
Units 1-5 RR at FG & 6-7 Existing FG	A	132.2	132.2	132.2	132.2	132.2	97.2	97.2
	B	119.0	119.0	119.0	119.0	119.0	87.5	87.5
	C	79.3	79.3	79.3	79.3	79.3	58.3	58.3
Units 1-5 RR at FG & 6-7 Existing BG	A	132.2	132.2	132.2	132.2	132.2	88.0	88.0
	B	119.0	119.0	119.0	119.0	119.0	79.2	79.2
	C	79.3	79.3	79.3	79.3	79.3	52.8	52.8

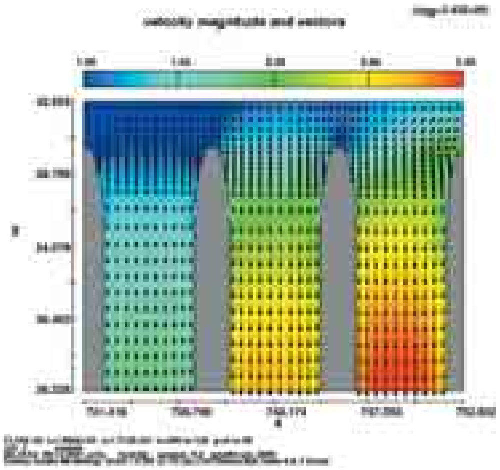
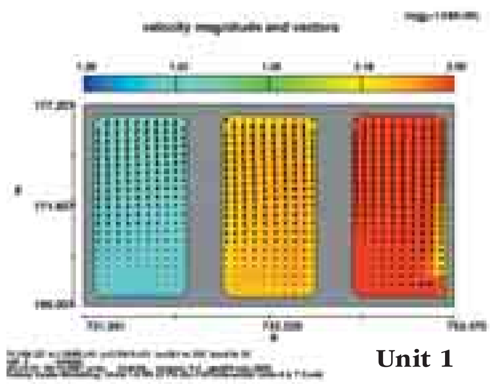


Figure 3: Velocity Distribution from Plan View and Profile View (Case 2)



Unit 1

program in 1990. Since there is no observed velocity data for the post re-runnering condition, only the three dimensional velocity plots are available.

Model Application

After the model was tested, it was applied to the following three scenarios:

Case 1: Units 1 to 7 re-runnering (RR) at full gate (FG)

Case 2: Units 1 to 5 re-runnering (RR) at full gate (FG) with units 6 and 7 at existing full gate (FG)

Case 3: Units 1 to 5 re-runnering (RR) at full gate with units 6 and 7 at existing best gate (BG)

The outflows used in the model for each case are presented in Table 2.

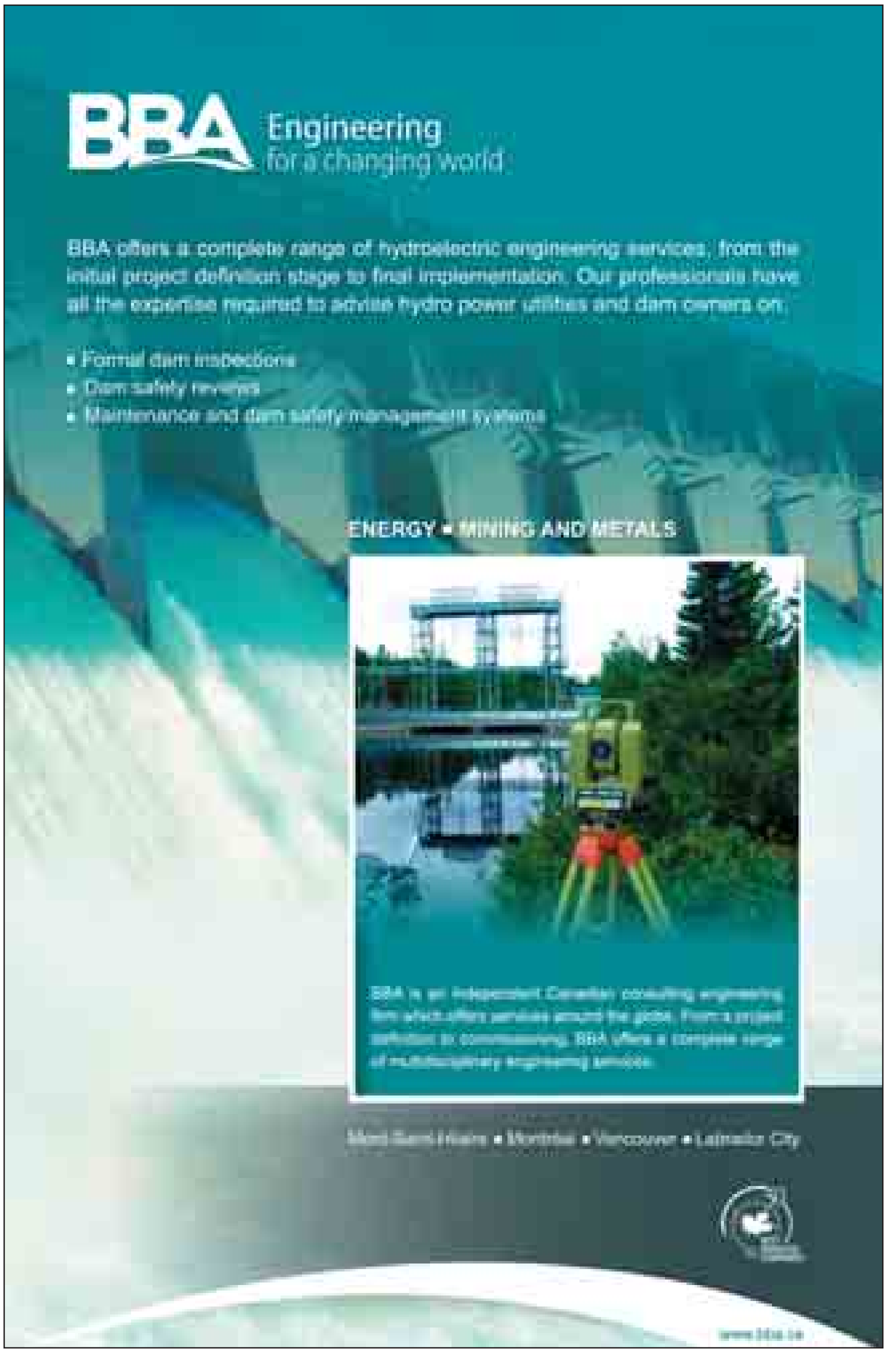
Model Results

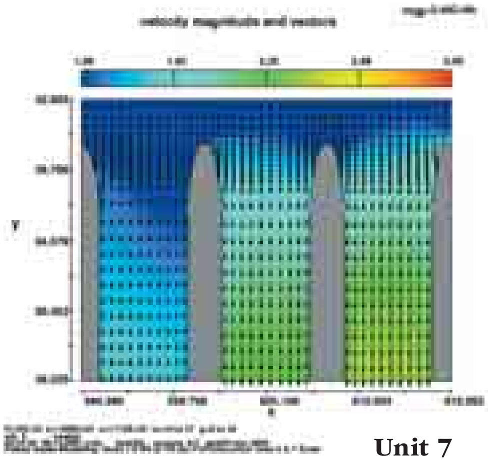
Velocity data was extracted along the X-Y plane of the model at elevation 161.5 m and also along the X-Z plane at a point located approximately 15 m upstream of each intake inlet. This information was provided to the manufacturer for their use in optimizing the turbine design. An example of the velocity results for Case 2 is presented in Figures 3 and 4. Figure 3 provides results for Unit 1, while Figure 4 provides results for Unit 7.

The results indicate that the velocity distribution is different between units in both the horizontal (X-Y plane) and vertical plane (X-Z plane) and most noticeable for units 6 and 7. This difference is caused by the presence of the rock knob and the change in flow direction just upstream of the powerhouse.

Conclusion

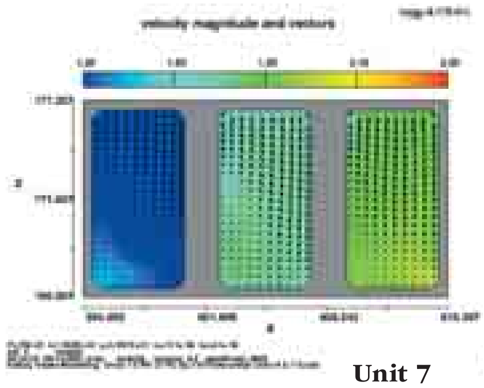
The FLOW 3D model of the intakes at the Kelsey Generating Station was setup and successfully used to model the velocity distribution through the powerhouse intake. This information was





Unit 7

Figure 4: Velocity Distribution from Plan View and Profile View (Case 2)



Unit 7

provided to the manufacturer and the information has been useful in helping to optimize the design of the turbines. ■

References

Efrem Teklemariam and Joe L. Groeneveld. 2000. Solving Problems in Design and Dam Safety with Computational Fluid Dynamics, Hydro Review, Vol. 5: 48-52.

Efrem Teklemariam, Brian W. Korbaylo, Joe L. Groeneveld and David M. Fuchs. 2002. Computational Fluid Dynamics: Diverse Applications in Hydropower Project's Design and Analysis, CWRA 55th Annual Conference, Winnipeg, Manitoba, CA, pp: 1-20.

Flow Science Inc. 2006. Flow 3D version 9.1 user manual.

Fuamba, M., Role. 2006. Behavior of Surge Chamber in Hydropower: Case of the Robert Bourassa Hydro Power Plant in Quebec, Canada, Dams and Reservoir, Societies and Environment in the 21st Century-Berga et al (eds) @ 2006 Taylor & Francis Group, London, ISBN 0415 40423 1.

Joe L. Groeneveld, Kevin M. Sydor, David M. Fuchs, Efrem Teklemariam and Brian W. Korbaylo. 2001. Optimization of Hydraulic Design using Computational Fluid Dynamics, Waterpower XII, July 9-11, Salt Lake City, Utah.

Marc St. Laurent, Efrem Teklemarian, Paul Cooley and Joe Groeneveld. 2002. Application of Hydraulic Models for the Environmental Impact Assessment of the Proposed Wuskwatim Generating Station, June 11-14, CWRA 55th Annual Conference, Winnipeg, Manitoba, CA

Michael C. Johnson and Bruce M. Savage. 2006. Physical and Numerical Comparison of Flow over Ogee Spillway in the Presence of Tail water, Journal of Hydraulic Engineering, Vol. 312, pp: 1353-1357.