Dams and Reservoirs, Societies and Environment in the 21st Century – Berga et al (eds)
© 2006 Taylor & Francis Group, London, ISBN 0 415 40423 1

Role and behavior of surge chamber in hydropower: Case of the Robert Bourassa hydroelectric power plant in Quebec, Canada

M. Fuamba
École Polytechnique de Montréal, Montreal, Canada

G. Brosseau, E. Mainville, S. Colameo & D. Sparks
Hydro-Québec, Montreal, Canada

ABSTRACT: This paper describes the Robert Bourassa surge chamber and its role in transient flow management. Using the *FLOW-3D*® numerical model, complex turbulent flow conditions in the surge chamber have been simulated, analyzed and compared to results obtained by a physical model study. A hydraulic-energy based method to determine head losses in the surge chamber is proposed, as well as identifying hydraulic conditions needed to optimize the production of electricity.

1 INTRODUCTION

The history of hydropower in Canada dates back to 1881, when the Ottawa Electric Light Company built a waterwheel plant at Chaudiere Falls to supply power for street lights and local lumber mills. In 1900, the first international transmission line between Canada and the United States was built across the border at Niagara Falls. By 1902, the Shawinigan Electric Company in Quebec had installed the largest generators in the world at Shawinigan Falls, and began sending power, at 50 kilovolts, some 135 kilometres to Montreal, along the longest transmission line in the world at the time.

Today, Canada has about 450 hydroelectric power plants in operation and more than 200 small hydro plants (which generate less than 3 MW of power). Among some of Canada's larger facilities are the Churchill Falls underground power plant in Labrador, the Robert-Bourassa (RB) complex near James Bay, Quebec and the 214 meter-high Daniel Johnson arch and buttress dam on the Manicouagan River in Quebec. Canada's installed hydropower capacity is 67,121 MW; its remaining technically feasible hydropower potential is 117,978 MW, the equivalent of 56 new Hoover Dams (USA), and twice the amount that is currently in operation. In Quebec alone, the installed capacity rose from 9700 MW to 32,000 MW in the last 40 years (Acore 2005).

The largest Canadian hydropower facilities not only produce electricity but have the capacity to store water for future energy production for days, months, or even years, depending on the size of their reservoirs. Often, in large underground plants, their facilities include surge chamber to prevent excessive pressure fluctuations in the draft tubes and to protect the turbine-generator units.

2 THE RB HYDROPOWER COMPLEX

2.1 *Description of facilities*

The La Grande River flows east to west, a distance of 800 kilometers before reaching James Bay. This river is the main tributary of Quebec slope of James Bay. It is the third largest river in Quebec with a drainage basin of 97,400 square kilometers, more than twice the area of Switzerland. The Robert Bourassa (RB) hydropower station, the largest of the eight hydropower stations on the La Grande River (LG-1, LG-2A, LG-3, LG-4, Laforge-1, Laforge-2 and Brisay) is located in the Municipality of Radisson, approximately 1,000 km north of Montreal.

All the RB facilities are underground. Water for the power station is retained by a dam creating a large reservoir of 61.7 billion cubic meters. Water is conveyed to sixteen turbine-generator units into the powerhouse through penstock pipes 40 m long. A surge chamber receives water from the turbines and directs it to all four tailrace tunnels. The surge chamber is located parallel to the power plant room for the machines. It was built to dampen strong pressure and waves fluctuations during unit starts and stops and flow variations.

It is 450 meters long, 45 meters high and 14 meters wide. It is divided into two half chambers which receive water from eight draft tubes and funnels this water to two tailrace tunnels. Figure 1 shows the profile view of the Robert Bourassa hydropower facilities.

Designed for a 10 000 year flood (16,280 cubic meters per second), a spillway has been provided to evacuate the water surplus of the reservoir during exceptional flood scenarios. It consists of 10 steps, each of 122 meters wide, 150 meters deep, and 10 meters high (see figure 2). The spillway's particular shape meets the environmental protection criteria. A tailrace canal of 1,500 meters long brings water to the river.

One RB turbine-generator unit produces 333 megawatts of power or 454,000 units of horsepower. This is equivalent to the power of three Boeings 747 during takeoff or 2,500 automobile motors. This is sufficient power to supply the Montreal subway or a city of 80,000 inhabitants.

2.2 *Role of the RB surge chamber*

Charge variations occurring in the turbine wheel, especially during the start-up, rate changes or the sudden shutdown, are sources of transient flow phenomena which propagate as pressure waves propagate towards hydraulic passages upstream and downstream. The waves can also reflect towards the perturbation source location and are reproduced in forms of amplified or reduced waves.

The role of the RB surge chamber is to reduce the surge value produced by the water hammer in the tailrace tunnels and to eliminate the surge produced by the water hammer in the draft tubes. The presence of the surge chamber helps to absorb the pressure fluctuations very quickly.

The surge chamber was optimized during design based on a physical model made in 1974 at École Polytechnique de Montréal. The physical model report contains useful information on the head losses as a function of the number of units and their configuration.

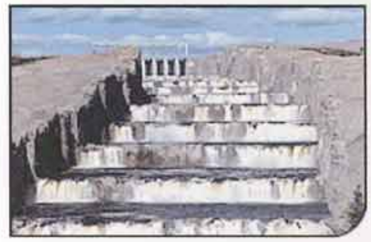

Figure 2: Frontal view of the RB spillway (Courtesy of Hydro-Québec ©)

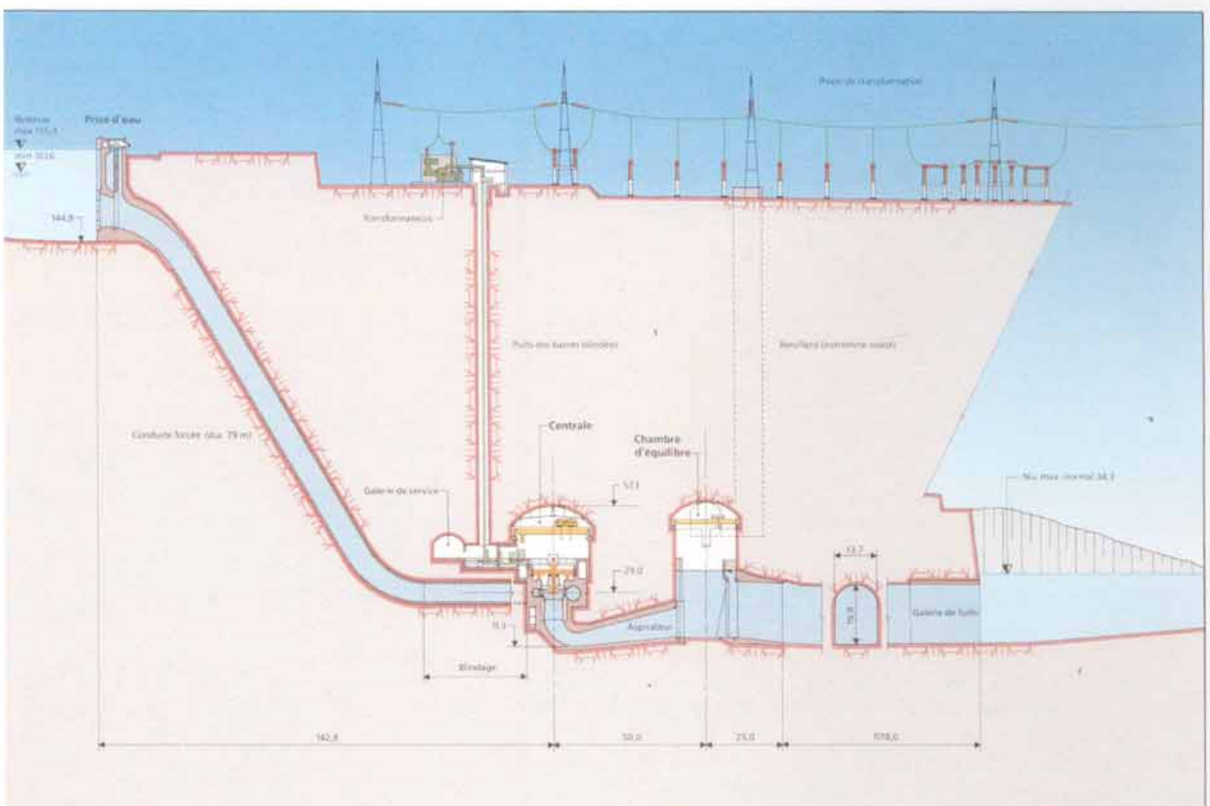

Figure 1: Profile view of the RB hydropower facilities (Courtesy of Hydro-Québec ©)

However only a few configurations had been tested and no longitudinal profiles in the surge chamber were produced (EPM 1974). Thus the goal of the study was to simulate the hydraulic behavior of the surge chamber and to analyze the impact of using different unit configurations on the chamber head losses.

3 NUMERICAL MODEL *FLOW-3D®*

3.1 *Numerical model characteristics*

FLOW-3D® is becoming one of most used 3D models in Computational Fluid Dynamics (CFD) Modeling. Based on the Navier-Stokes equations it uses a fractional area/volume method (called FAVOR™) for modeling complex geometric regions. In this method, the width of the open portion of the cell is equal to the product of the open volume fraction and the original cell width. This product is used for difference approximations in the horizontal direction. All equations are formulated with area and volume porosity functions. For example, zero-volume porosity regions are used to define obstacles, while area porosities may be used to model thin porous baffles. Porosity functions also introduce some simplifications in the specification of the free-surface and wall boundary conditions (Flow Science 2005).

The model enables simulation of rapidly varied flow, erosion and deposition as well as other types of simulations. It can consider the bi-phase flow (liquid/gas, two liquids of variable densities, stratified fluids), and cavitation. It can use different turbulence models like K-ε, Large Eddy Simulation (LES) and Prandtl Mixing Length. This model has been used as complement or alternative to physical modeling in hydropower projects for dam break studies, derivation canal optimization, spillway design, hydraulic capacity assessment, …(Joannette et al. 2004, Ho and al. 2003, Hirt and Nichols 1981, Teklemariam and al. 2002, Savage and Johnson 2001, Ho and al. 2001).

3.2 *Numerical model application*

To apply the model one must obtain the 3D geometry of the hydraulic structures as well as boundary and initial conditions of the flow. This geometry is then used to generate a grid, done by the software with a drawing feature. The numerical model uses an orthogonal mesh defined in terms of either cartesian or cylindrical coordinates. Complicated geometries can be modeled. Therefore obstacles and baffles are embedded in the orthogonal mesh by partially blocking cell volumes and faces areas. This allows independent definition of the mesh and geometry, i.e. the geometry may be modified without redefining the mesh. Mesh generation is much simpler and faster than it would be for body fitted coordinates. The mesh is defined independently for each of the three orthogonal coordinates (Flow Science 2005).

3.2.1 *Setting of the RB 3D geometry*

Three dimensional geometrical components were created. The first one represents the horizontal part of the draft tubes, the second one the inclined part and the third one the surge chamber along with the two tailrace tunnels. All the details of the surge chamber were reproduced according to the RB 'As Built' drawings.

Three mesh blocks were created in accordance with the degree of accuracy needed. The first block includes the upstream reservoir and all the draft tubes. A reservoir was inserted at the entry of the draft tubes to serve specifically as an upstream boundary condition to the system. The grid mesh related to the first block measures 50 m in length, 200 m in width and 70 m in height. The related mesh resolution is of 1.0 m x 1.0 m x 1.0 m with 840 000 cells.

The second mesh block starts 4 m upstream of the draft tubes entrance. It is 28 m long (4 m downstream of the surge chamber exit), 200 m wide and 70 m high. The mesh in this region must be fine in order to accurately to represent the complex phenomena such as turbulence and vortex in this zone under study. More density in the layer limit zone can introduce and intensify numerical errors. As suggested by Wilcox (Flow-Science 2000), mesh should not be more dense, but just enough to cover the region where the turbulence effect is important. The suggested ratio in elevation (z) between the optimal cell height and the useful depth of the canal is 8%. Starting with the same ratio, the dimension z of the cells was optimized to 0.80 m according to the convergence tests. Finally the mesh resolution in the second block is 1.0 m x 0.8 m x 1.0 m with 490 000 cells.

The third block includes tailrace tunnel # 1 and half of the downstream reservoir. It starts 10 m downstream of the tunnel entrance and is 500 m long, 20 m wide and 56 m high; the related mesh resolution is 1.0 m x 6.7 m x 1.1 m with 100 000 cells. The fourth block includes tailrace tunnel # 2 and the second half of the downstream reservoir. Its dimensions are identical to the third one, like the mesh resolution and the number of cells. A dense mesh is not required for blocks 3 and 4 because these zones are out of interest.

3.2.2 *Basic hypothesis*

An absolute roughness value of 100 mm is considered to represent internal tunnel section (Graf and Altinakar 1998). Water is incompressible and its dynamic viscosity value is 0.001 N.s/m^2. The k-ε model is used because it gives the best approximation of the dynamic flow conditions with turbulence.

Boundary and initial conditions

Small upstream and downstream reservoirs are provided to set the boundary conditions. A discharge whose value varies between 705 and 2319 m^3/s is considered as an upstream boundary condition. This condition is set into the numerical model in terms of velocity which is equally distributed throughout the draft tube cross sections. A water level in the downstream reservoir is considered as a downstream boundary condition.

Initial conditions are defined by zero velocity values. At the upstream boundary, the velocities increase progressively during numerical simulations to reach the fixed value corresponding to the turbine discharge.

4 DESCRIPTION AND ANALYSIS OF FLOW CONDITIONS IN THE RB SURGE CHAMBER

4.1 *Introduction*

A few numerical simulations were conducted to validate the numerical model and to analyze the flow conditions into the RB surge chamber. Simulations were performed until the steady state (or the stability limit of the model with value of 5.7 E-02) was reached in the system. Simulations were time consuming because of the RB geometry, the number of the mesh cells and the complexity of the flow system. A 500 second simulation takes a CPU time of 25.92 x 10^4, 3 days at least on a Pentium 4 computer. Due to the fixed water level at the downstream boundary and the fixed discharge at the upstream limit, the numerical model will determine the water level values at the upstream boundary and into the surge chamber. The value in the surge chamber is compared to the observed value. If the computed value is not similar to the observed one, the value of the downstream water level is revised and a new simulation is run.

4.2 *All units open, $Q_{obs.}$ = 2,319 m^3/s*

Simulations have shown that a portion of water flow from draft tubes # 1, 2, 4, 5, 7 and 8 crosses the surge chamber before being deviated towards the two tailrace tunnels. These water masses deviate when approaching the downstream wall of the chamber. The flow turns into a helicoidal type, as shown in Figure 3 where a low velocity zone (0.8 m/s) can be seen in the center of the surge chamber above the elevation of 47 m.

Flow of several trajectories crosses the surge chamber zone located in front of draft tubes # 3 and 6. First of all, all the flow from draft tubes # 3 and 6 crosses the surge chamber and get directly into the tailrace, as shown by Figure 4. The water masses from closer draft tubes approach the considered zone and are directed to the tailrace tunnels by the lintel wall (small wall in front of the tailrace entrance). The predicted entrance velocity is on the order of 3.75 m/s at the lintel wall elevation and 5 m/s close to the surge front invert. Water masses above the lintel wall are reflected on the surge chamber downstream wall, provoking a reverse flow which will be sucked in near the tailrace tunnels, when propagating towards the upstream wall.

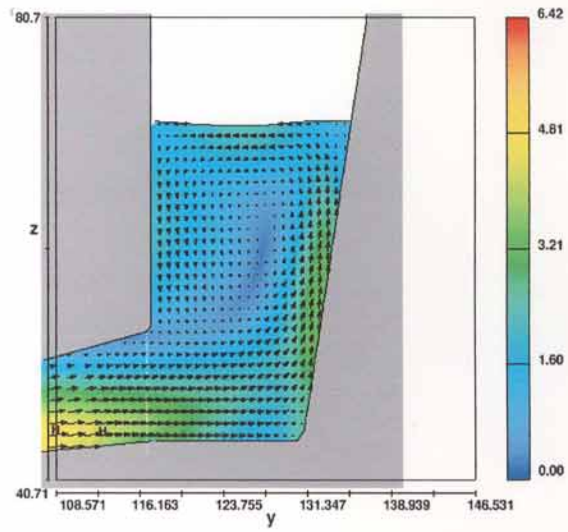

Figure 3: Flow of helicoidal type from draft tubes # 1, 2, 4, 5, 7 and 8 in the RB surge chamber

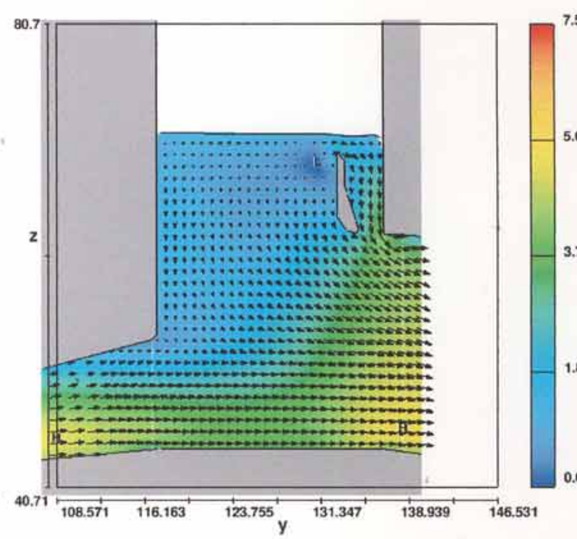

Figure 4: Flow from draft tubes # 3 and 6 in the RB surge chamber

4.3 *Three units off, five units functioning*

Simulating the case study where the three units (3, 5 and 6) are off while the other five are discharging 1 238 m^3/s shows that all the water masses from the turbine-generator units 1 and 2 together with 7 and 8 will pass through tailrace tunnels 1 and 2 respectively (see Figure 5). The water flow of the turbine-generator unit 4 is divided between the two tailrace tunnels. The predicted discharge distribution is 49% in tailrace 1 and 51% in tailrace 2.

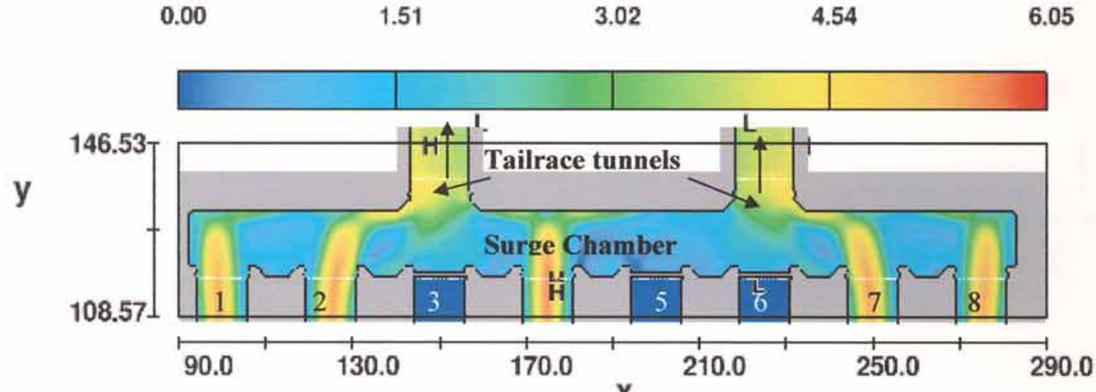

Figure 5: Flow conditions in the RB surge chamber when 3 units are off

5 HEAD LOSSES THROUGH THE RB SURGE CHAMBER

5.1 *Simulated case studies*

Seven configurations were simulated. The first is when all the turbine-generator units are functioning at full capacity. The second to fifth cases are those where three units are shut off and in the last two studies, only three units are functioning. Table 1 shows that the difference between the values of the discharge predicted by the numerical model and the ones measured for the seven considered case studies do not surpass 1.4%. The difference between the predicted and observed water level values in the surge chamber were maintained inferior to 16 cm, except for two cases: # 3 and 7.

5.2 *Head losses and energy prediction*

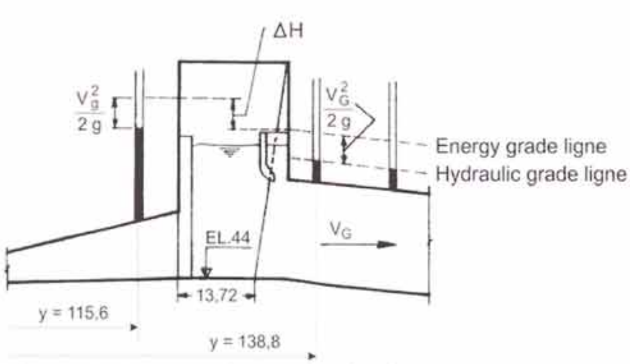

Figure 6: Cross sections wherein the average energy values are calculated

The cross sections wherein energy values are calculated are represented in Figure 6. The terms V_g and V_G represent the average velocity in the cross sections respectively of draft tubes and tailrace tunnels. First of all the energy is calculated in the defined cross sections using the Bernouilli equation.

The used numerical model gives values of velocity U (in the direction x), V (in the direction y), W (in the direction z) and P (the local pressure). The head loss is determined by taking the difference between two calculated energy values.

Then the head loss coefficient k is computed by using the average draft tube/tailrace cross section velocity V and the head loss value ΔH in the following equation $k = \Delta h/(V^2/2g)$

Table 2 shows that the head loss values do not surpass 65 cm in the surge chamber for an average discharge of a turbine-generator unit which limited to 290 m^3/s, high losses for high total flow. The lowest loss (23 cm) in the surge chamber is found with a discharge value of 235 m^3/s. The losses in the cross section located 4 m downstream of the entrance of the surge chamber remain limited between 21 and 30 cm. That could be explained by the fact that the major head losses are singular losses which occur at the surge chamber entrance. The friction head losses seem to be slighter.

When examining the case studies where a couple of turbine-generator units are shut off, table 2 shows that substantial losses occur with the stopping of turbine-generator units 3 and 6 simultaneously. This is configuration # 3 where the head loss coefficient is the highest one for the configurations with three units off.

Table 1 : Discharge and water levels values in the surge chamber during the steady state regime

Case No	Unit shut off	Q (m^3/s) Observed	Q (m^3/s) Predicted	Q Diff. (%)	Water level (m) Observed	Water level (m) Predicted	Water level Diff. (cm)
1	0	2,319	2,328	-0.4	41.19	41.34	15
2	2 - 5 – 6	1,271	1,264	0.6	34.67	34.70	3
3	3 - 5 – 6	1,238	1,235	0.2	35.32	34.80	-52
4	1 - 4 – 7	1,196	1,212	-1.3	34.29	34.27	-2
5	4 - 5 – 6	1,183	1,166	1.4	35.06	65.07	-1
6	1 - 2 - 3 – 4 - 8	705	699	0.9	34.24	34.17	-7
7	1 - 2 - 4 – 6 - 8	705	708	-0.4	34.24	33.64	-60

Table 2 : Head loss values in the surge chamber

Case No	Unit shut off	Q_{ave}/DT (m^3/s)	ΔH_1 (m)	k_1	ΔH_2 (m)	k_2
1	N/A	290	0.65	0.62	0.30	0.29
2	2 - 5 - 6	254	0.39	1.28	0.25	0.81
3	3 - 5 - 6	248	0.48	1.65	0.25	0.84
4	1 - 4 - 7	239	0.26	0.92	0.23	0.81
5	4 - 5 - 6	237	0.37	1.41	0.21	0.80
6	1 - 2 - 3 - 4 - 8	235	0.27	2.85	0.25	2.63
7	1 - 2 - 4 - 6 - 8	235	0.23	2.42	0.22	2.33

Notes :

(1) Q_{ave}/DT means the average discharge of a draft tube.

(2) Subscripts 1 and 2 refer to cross sections which are downstream and in the surge chamber (y = 121.2 m) respectively.

Table 3 : Choice of the optimal configuration when using five turbine-generator units Solely from a hydraulic point of view

Choice	Units shut down	Q (m^3/s)	k_{1c}
1	1 - 4 - 7	1,200	0.93
2	2 - 5 - 6	1,200	1.27
3	4 - 5 - 6	1,200	1.41
4	3 - 5 - 6	1,200	1.88

6 OPTIMAL UNITS MANAGEMENT

For a specific number of required running units there is an optimal hydraulic configuration in the surge chamber. When the discharge is in the order of 1,200 m^3/s, it is best to have five units running. Table 3 indicates head losses in the chamber for four situations with five units running. The head losses are minimized while units 1, 4 and 7 are closed. On the other hand, they are maximized when units located opposite to the two tailrace tunnels (3 and 6) are shut down.

Two situations have been simulated when three units are requested. Configurations 6 and 7 show that case 7 should be preferred to case 6 because of the lower head loss coefficient (2.33 versus 2.63).

7 CONCLUSION

The RB hydropower facilities have been described and the role of the surge chamber pointed out in this paper. Simulations have been performed with the numerical model *FLOW-3D*® to validate the observed results obtained from a physical model representing the RB surge chamber and to predict flow conditions in the chamber. Predicted results agree very well with observed ones.

Even the water surface into the surge chamber is not flat during the steady state regime, numerical results show that the water level variation is slight, between 10 and 20 centimetres at full flow regime, when all units are running. Substantial losses occur with the stopping of turbine-generator units 3 and 6 simultaneously both when 3 or 5 units are running.

A hydraulic-energy based method to determine head losses in the surge chamber has been proposed, as well as a method to manage the opening of units which will optimize the production of electricity.

ACKNOWLEDGEMENT

Hydro Quebec Production provided the financial support for this project. The contribution of the 'Expertise de centrales' Unit is acknowledged.

REFERENCES

ACORE 2005. The American Council on Renewable Energy-Web Site: www.acore.org

EPM 1974. *Aménagement hydroélectrique LG-2, chamber d'équilibre du système de fuite, étude sur modèle réduit.* Technical report produced by École Polytechnique de Montréal with RSW Engineering Consulting Group and submitted to Société d'énergie de la Baie James, Montreal, Canada.

Flow Science Inc. 2000. FLOW-3D®: *Modeling Roughness Effects in Open Channel Flows.* Technical report FSI-02-TN60. USA.

Flow-Science, 2005. *FLOW-3D User's Manual*, version 9.0, Excellence in Flow Modeling Software, The FAVOR advantage. USA pp. 587.

Graf, W.H. and Altinakar, M.S. 1998. *Hydrodynamique: une introduction.* Presses polytechniques et universitaires romandes. Lausanne, Suisse p. 437.

Hirt, C.W. et Nichols, B.D. 1981. Volume of fluid (VOF) Method for the dynamics of free boundaries. *J. Computational Phys.*, 39, 201-225.

Ho, D., Boyes, K., Donohoo, S. et Cooper, B. 2003. Numerical Flow Analysis for Spillways. 43rd *ANCOLD Conference*, Hobart, Tasmania.

Ho, D., Boyes, K., Donohoo, S. 2001. Investigation of Spillway Behaviour Under Increased Maximum Flood by Computational Fluid Dynamics Technique. *Proc. Conf. 14th Australian Fluid Mechanics*, December 2001. Adelaide, Australia, 577-580 pp.

Joannette, J., Quach, T., Fuamba, M. et Mercier, J.F. 2004. *Simulation numérique des écoulements en 3D: Cas des barrages Mercier et Les Cèdres au Québec. CDA 2004 Annual Conference*, September 25 – October 1, 2004, Ottawa, Ontario, Canada.

Savage, B.M. and Johnson, M.C. 2001. Flow over Ogee Spillway: Physical and Numerical Model Case Study. *Journal of Hydraulic Engineering*, ASCE. 640-649 pp.

Teklemariam, E., Korbaylo, B.W., Groeneveld, J.L., Fuchs, D.M. 2002. Computational Fluid Dynamics: Diverse Applications in Hydropower Project's Design and Analysis, *CWRA 55th Annual Conference June 11 – 14*, 20 pp.