Numerical and physical modeling of hydraulic structures

Hydraulic structures are used to control the flow of water in hydropower developments, urban drainage and irrigations projects. Many of these open channel flow design problems are unique and have complex boundary conditions due to their site specific geometric design. Since they are insoluble by theory or by reference to standard empirical data, their solution is found either by physical hydraulic modeling or, more recently, by numerical modeling. There is a long experience in physical scale modeling which allows for the Froude model results to be interpreted by the engineer to aid in the prediction of prototype behavior. As computing power is increasing, the more recent numerical models are offering an alternative. However for large complex problems, before results are used they must be validated as constraints due to time, computing power, input boundary conditions and model limitations, cause approximations to be used resulting in the flow being inadequately modeled.

Projects pursued are usually motivated by particular design problems and are done in direct collaboration with industry.

Current Project: *Free surface vortices at hydropower intakes*: Free surface vortices may significantly reduce turbine efficiency and cause premature mechanical failure when they occur at hydropower intakes so preventing their occurrence is a priority during the design and optimization of an intake. This optimization process is particularly important for low-head intakes, where the presence of free surface vortices is more frequent. Through predictive correlations exist; their reliability is limited to specific geometries, so when a high level of confidence is required in the quality of the flow conditions in a proposed intake, the current engineering practice is to construct a physical model of the intake to ascertain that no strong vortices will occur under the design operating conditions.
Computational fluid dynamics (CFD) software has presented itself in recent years as an appealing alternative to physical scale modeling with the potential to provide detailed velocity field data at a lesser cost than physical modeling. However, before CFD can be used with any confidence to predict vortex formation at intakes, it must be validated using experimental data about approaching flow conditions and corresponding vortex intensities. A number of challenges must also be overcome in applying a given CFD code to hydropower intake flow, including resolving the large range of physical scales, capturing the free surface deformation and efficiently modeling turbulence in the presence of vortices.

Past projects: *Head loss in flow past fish louvers:* Fish louvers are used at dams to deflect the fish away from the hydropower intakes and towards a fish bypass. Head loss past the louvers was investigated for several louver designs. *Spillway aeration:* Spillways are used to pass excess water past a dam and for high head dams, the flow down the spillway can reach such high velocities that cavitation occurs. Cavitation, which results in erosion of the spillway face, can be prevented by aerating the spillway flow. Spillway aeration was physically modelled and an empirical prediction was developed. *Numerical modelling of flow through a spillway and diversion*: a commercial CFD model was used to model the flow through a spillway and diversion and the results compared to the physical scale model. The numerical model performed reasonably except for cases with rapidly varying flow.

<ins>Collaborators</ins>:
Graham Holder & Marc Villeneuve, Groupe Conseil LaSalle, Canada
Dr. Etienne Parkinson, VATech, Switzerland
IREQ, Montreal, Canada
ANSYS Canada

Student theses:
Current:

France Suerich-Gulick (Ph.D.): Complex open channel flows at hydro power intakes. (Collaboration with Groupe Conseil LaSalle, VATech, IREQ, Ansys)

Morrisette, David (M.Eng.): Physical modelling of hydropower intakes.

Past:

Gacek, Julian (M.Eng. 2007): Numerical simulation of flow through a spillway and diversion. (Collaboration with Groupe Conseil LaSalle)

Tarek Hamade (M.Eng. 2004): An experimental study on a fish louver system. (Collaboration with Groupe Conseil LaSalle)

Aubel, Tristan (Visiting Research Student from ENSEEINT (INP Toulouse, France) Aerated flow on spillways with a ramp offset aerator.

Publications:

Suerich-Gulick, F., Gaskin S., Villemeuve, M., & Parkinson, E.(2007,) Numerical simulation of a free surface vortex formed in the wake of a pier at a hydropower intake, *32nd Congress of IAHR*, July 1-6 2007, Venice, Italy, pp.1-10.

Suerich-Gulick, F., Gaskin S., Villemeuve, M., Holder, G. & Parkinson, E. (2007 extended abstract, poster) Impact of geometry on free surface vortices in a scale model of a hydropower intake, *Hydraulic Measurements and Experimental Methods, EWRI & IAHR*, Lake Placid, New York, Sept 10-12, 2007.pp 1-5.

Suerich-Gulik*, F., Gaskin, S., Parkinson, E., Villeneuve, M., Holder, G. (2006) Experimental and numerical analysis of free surface vortices at a hydropower intake, Proceedings of the *7th International Conference on Hydroscience and Engineering, ICHE 2006* [ISBN: 0977447405], http://hdl.handle.net/1860/1414. Philadelphia, USA, Sept. 10-13, 2006.pp.1-11.

Suerich-Gulick, F., Gaskin, S. Parkinson, E. & Villeneuve, M. (2005) Free surface simulation of hydraulic turbine intakes, *23rd CADFEM Users' Meeting 2005,International Congress on FEM Technology with ANSYS CFX & ICEM CFD Conference*, Bonn, Germany, 9-11 Nov., 2005. pp. 1- 8.

Gaskin, S., **Aubel, T.** & Holder, G. (2003) Air demand for a ramp-offset aerator as a function of spillway slope, ramp angle and Froude number, XXX IAHR Congress, Thessaloniki, Greece, 24-29 August, 2003. *Water Engineering and Research in a Learning Society: Modern Developments and Traditional Concepts, Theme D*, pp.719-726.