Hydraulic Simulation with 3D Computational Fluid Modelling in a Flood Event

Eric Chen, P.Eng., M.Eng., PMP,
Glen Ichikawa, P.Eng.

March 2019

BBA.CA

Disclaimer

This document is provided for informational purposes only. Anyone who uses this document is responsible for determining whether or not it applies to his or her activities. At no time will BBA or the author be held liable for any error or omission that the document may contain or that may result from its use.

The information contained in this document is the proprietary and exclusive property of BBA Inc. except as otherwise indicated. No part of this document may be reproduced, stored, transmitted, or used for design purposes without the prior written consent of BBA Inc.

The information contained in this document is subject to change without notice.

Executive Summary

The concept of computational fluid dynamics (CFD) and its applications can date back as early as the 1960s. Over the years, due to the development of both theoretical research and computer technologies, design engineers and researchers have found CFD to be a very useful tool for complex problems where manual calculations are neither sufficient nor possible.

This paper reveals an example of CFD modelling in Ansys CFX to simulate a 200-year return period for a river flood event at a powerhouse site where there are combined effects of a hydraulic jump at a wall fall and water backup by a narrow canyon. This paper also introduces the model assumption and methodology, explains meshing technology and boundary conditions and reviews model results. In addition, it covers model verification techniques.

Although this paper is an example of applying CFD in a river flood simulation, a similar methodology and concept have been applied in many other applications and industries, which are listed at the end of this paper

Table of Contents

Introduction

The powerhouse for a hydropower project is usually located as low as possible to generate more energy. However, the powerhouse floor elevation is also limited by various constraints, like flood levels. It is very important to calculate river elevations during a flood. This calculation has to be accurate, as it relates directly to project revenues. Traditionally, manual calculations or a 1D/2D numerical model (e.g., HAC-RAS) is used to do this; however, these calculations are sometimes not as accurate and do not provide sufficient flow performance information when combined with complicated situations.

The example simulation described in this paper is for a 1:200 year return period for a river flood. The proposed powerhouse location is downstream from a waterfall and upstream of a long narrow valley (see Figure 1). Three hydraulic phenomena need to be considered in order to assess this problem:

- Hydraulic jump and wave run-up are expected at the pool after the waterfall due to momentum and kinematic energy.
- Water backup is expected at the pool due to the narrow valley downstream.
- High turbulence is expected at the pool due to a 90o turn in the river, which alsocontributes to the first two phenomena.

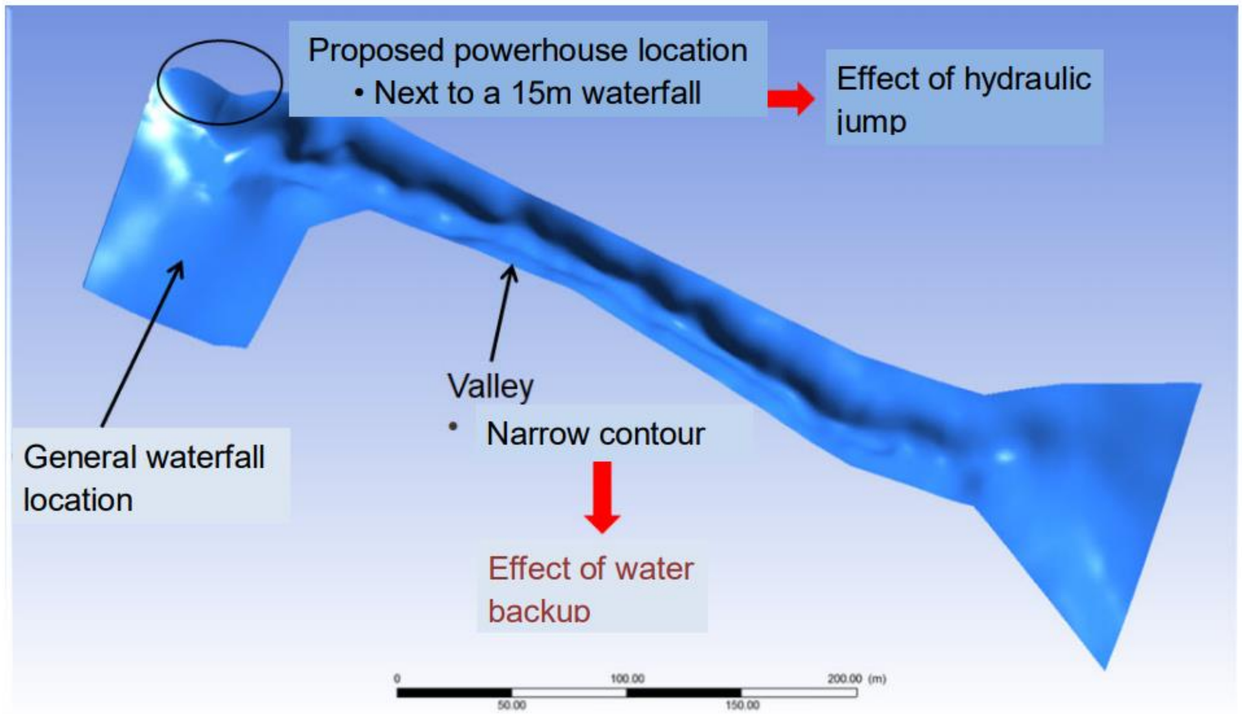

Figure 1: Model geometry

Because of the above-mentioned factors, a 3D numerical model is the most efficient solution for this type of application, as it can produce detailed and reliable results within a relatively tight timeline. It is also much more cost effective than physical modelling in both the preliminary and detailed design stages.

Numerical Model Simulation

3D model geometry

The model geometry is created in SolidWorks and based mostly on the site topography.

- A 3D laser scanner is used to travel along the waterfall and canyon to capture the true 3D riverbed geometry and generates around 36 billion survey points.
- The survey points are post-processed to remove any noise.
- The processed survey points are then combined with original site topography (upstream of the waterfall) to create the 3D geometry.

Once created, the model geometry is imported into Ansys for the meshing process. A proper mesh methodology is chosen for this geometry and creates about 18 million elements. Mesh refinement is also carried out to apply finer mesh in the critical areas for more details.

Mesh adaption technique

Note that for this example, the focus is to simulate flood elevations to localize the powerhouse, so it is very important that the free surface location be accurate. Therefore, in addition to localized mesh refinement, a mesh adaption technique is employed in this model for the free flow surface. This technique basically sets the adaption criteria so that when the initial water-free surface location is calculated by the solver, elements within a certain range of this surface will be refined automatically. The solver will then re-run the model to obtain a more accurate free surface location.

This will take three iterations; each iteration generates more detailed and reliable results.

Model setup and boundary conditions

This simulation uses a 2-phase hydraulic model (i.e., air and water)—the Eulerian-Eulerian Model. This model is commonly used to mix and separate phases. It solves momentum, enthalpy and continuity equations for each phase and tracks volume fractions. Both water and air in this model are assumed to be Newtonian, incompressible fluids.

The K-Epsilon turbulence model is used in the simulation for flow turbulence performance. This model is the most widely used engineering turbulence model for this type of application. Various papers and researches have proven that this model is stable and robust enough for this type of analysis. It follows the eddy viscosity concept (EVM), which has a built-in isotropy assumption. This means that the turbulent velocity is assumed to be equal in all directions.

A scalable wall function is added to this turbulence model to increase model accuracy for near wall/edge turbulence behaviour.

Figure 2 shows the boundary conditions of the model.

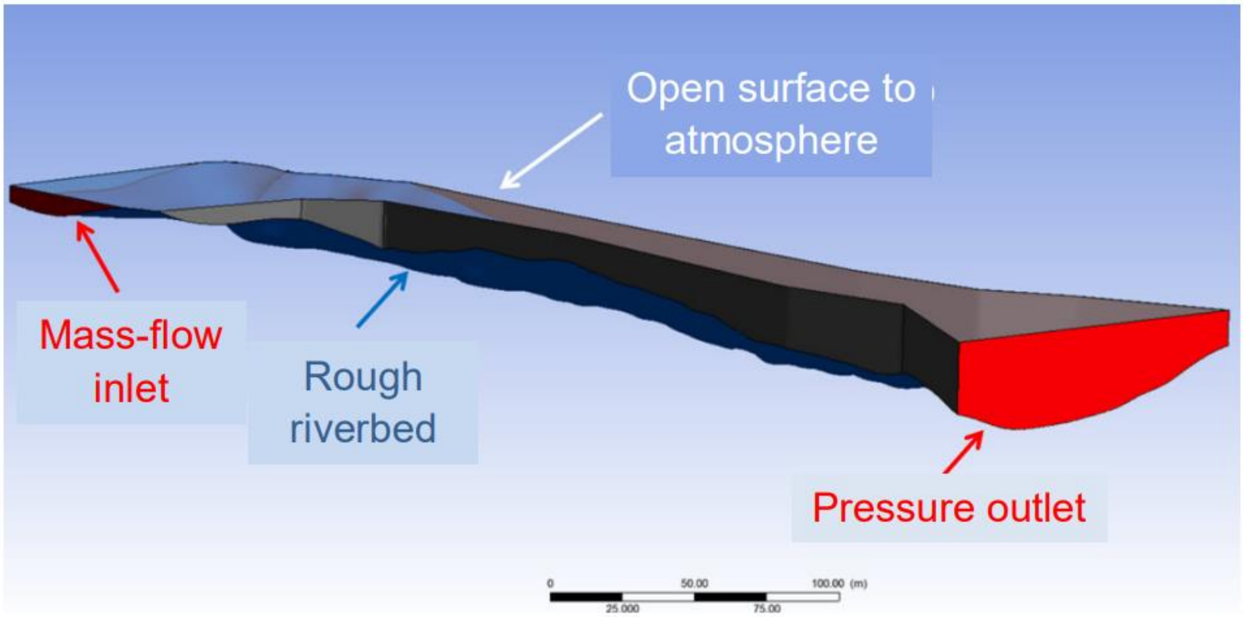

Figure 2: Boundary conditions

Model Results

Figures 3 to 6 show model results. The hydraulic jump occurs at the pool and significantly high waves run up the bank on the opposite side of the waterfall (Figure 6), as expected. Turbulence and eddies form at the pool and the velocity field is simulated. The water velocity accelerates at each narrow location along the canyon and also represents water backup upstream of those locations.

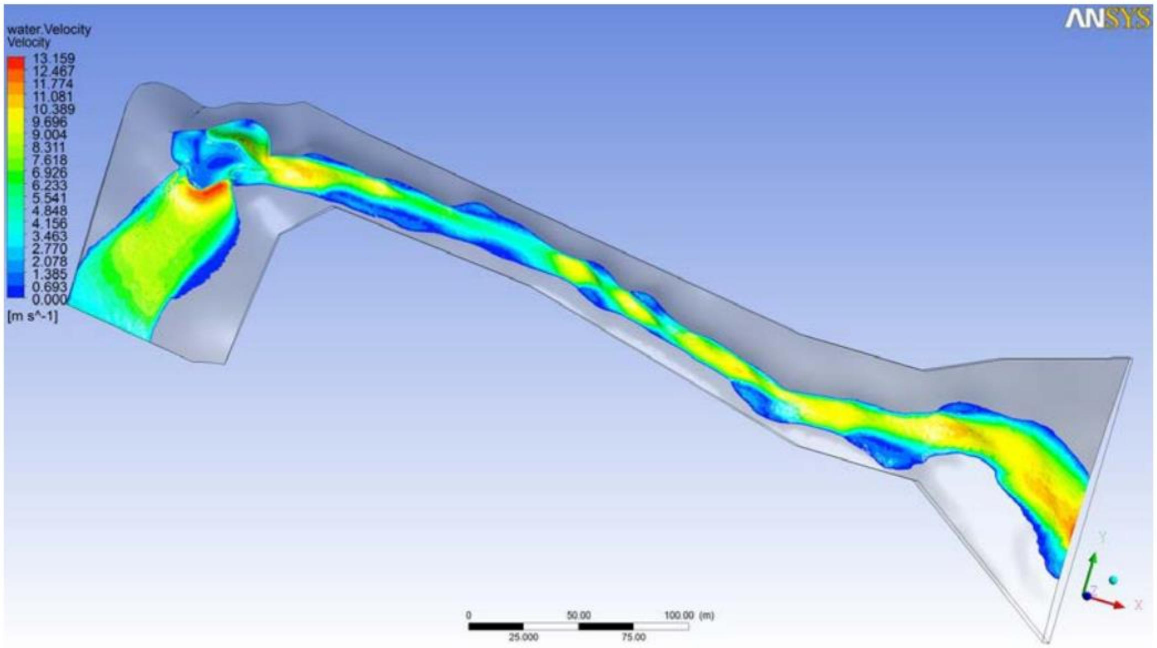

Figure 3: Overall flow velocity at the free surface

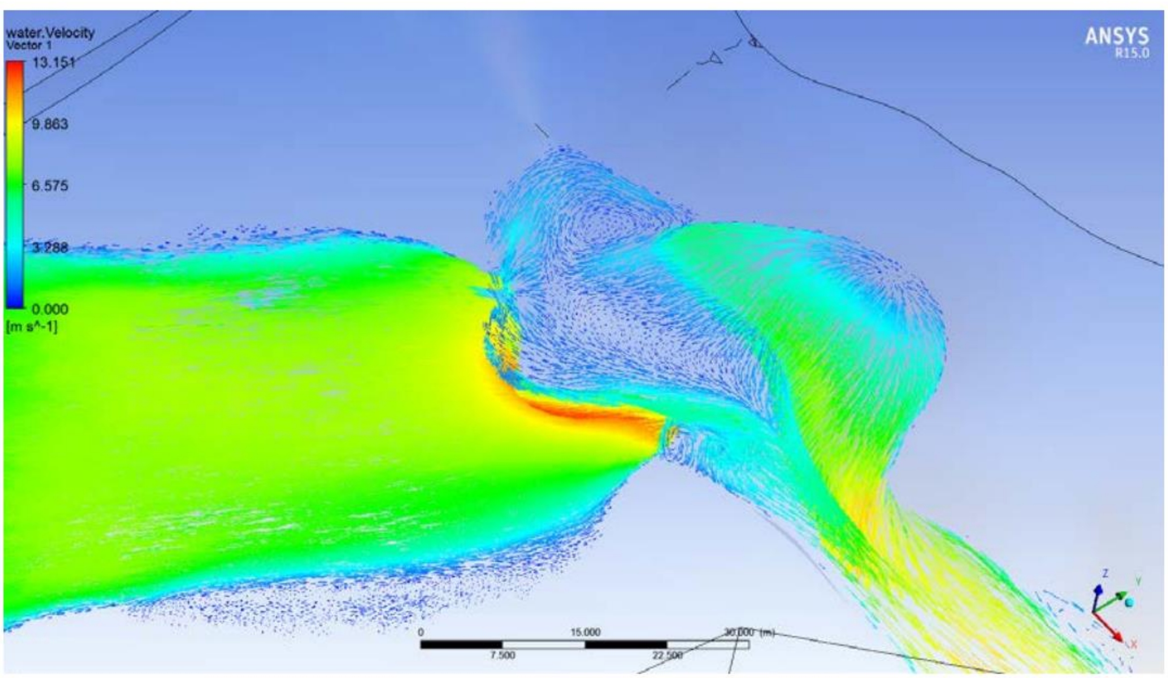

Figure 4: Velocity vector at the waterfall

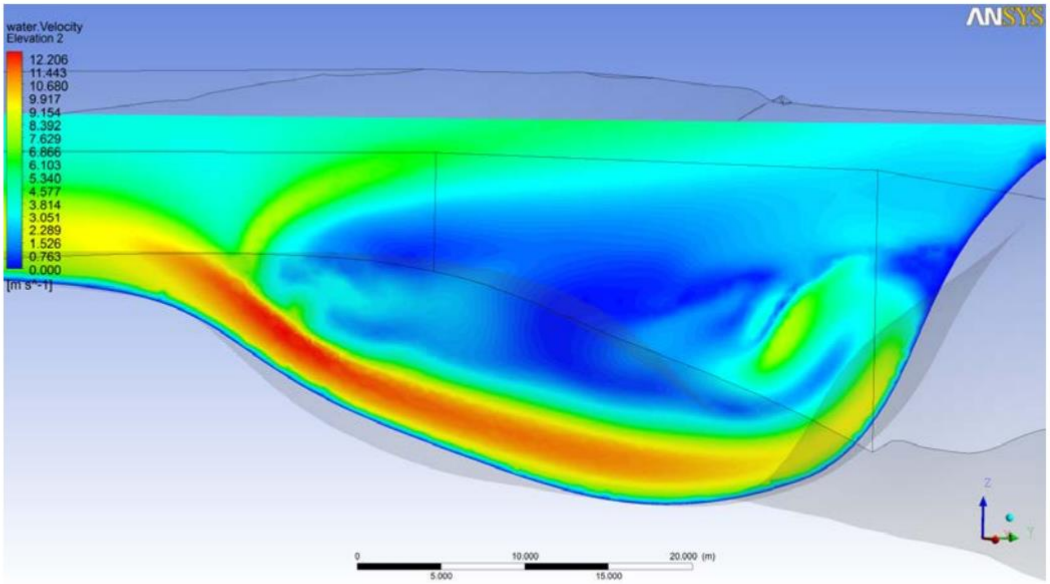

Figure 5: Section view of the waterfall to the powerhouse (velocity field)

Figure 6: Free surface elevation from the waterfall to the powerhouse

Model Validation

Grid independent analysis

Mesh quality is important to model accuracy and convergence, especially for multi-phase models that have many variables and equations. Therefore, a grid independent analysis is carried out by checking the model without changing parameters, except the meshing methodology and element size. The comparison shows consistency in the results among different mesh cases.

The element mesh quality was also checked against orthogonality, aspect ratio, expansion rate and other criteria.

Model sensitivity analysis

A series of sensitivity analyses are conducted by changing one input parameter, without changing the rest of the parameters or meshing methodology (see below). The results are compared with the base model for differentiation.

- Turbulence models

  A separate simulation using the Menter shear stress transport (SST) turbulence model is created to compare with the base case (i.e., k-ε turbulence model) for pressure distribution, velocity field, turbulence kinetic energy, etc.

- Riverbed roughness

  The riverbed is subject to weathering and debris/sediment deposition, which will smooth or roughen the riverbed over the time. Thus, several sensitivity cases are carried out with varying riverbed roughness.

- River channel blockage

  During the flood event, the river flow can be blocked by fallen trees or other large objects on the river's narrowest locations, which could potentially back up the water and increase water elevation. As a result, a series of separate models are created to simulate river channel blockage at different locations (see Figures 7 and 8) and the effects they have on water elevation at the powerhouse location are evaluated.

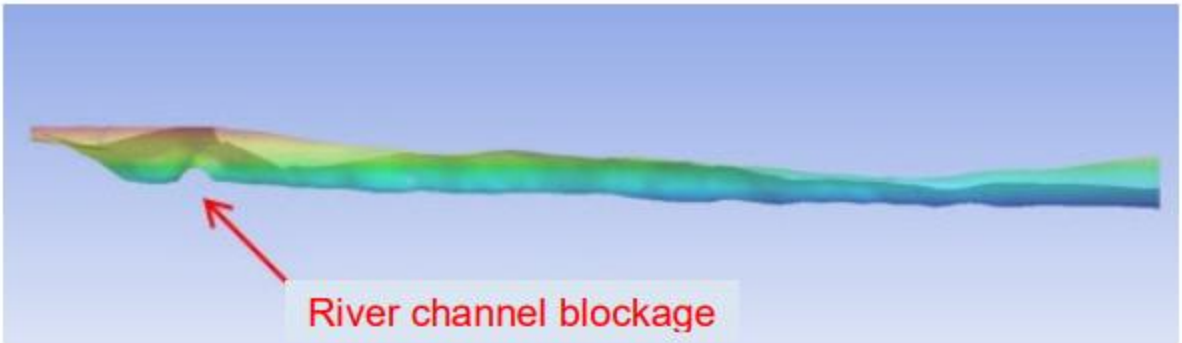

Figure 7: River channel blockage (1)

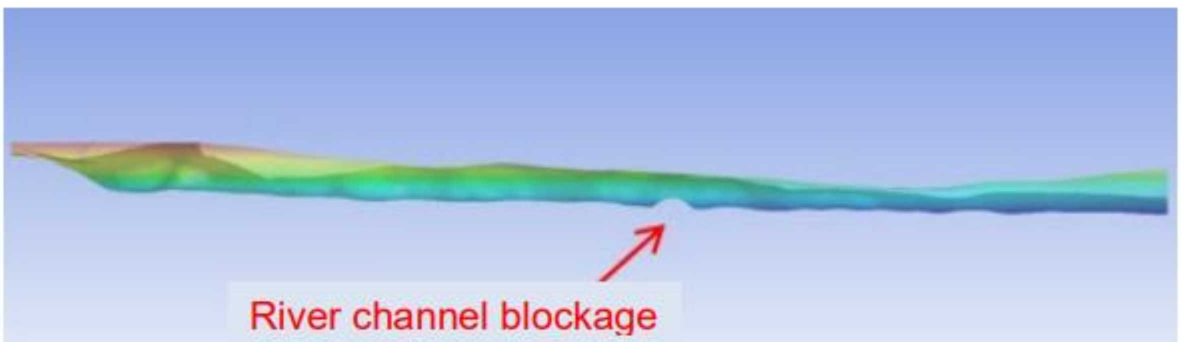

Figure 8: River channel blockage (2)

- Model results vs. onsite measurement

  A separate model with 1:10 year flood flow is created with the same mesh and model inputs. The model results are placed side by side with flow measurements that are available for a 10-year period flood event at this site. The comparison shows that the model results are aligned with real-life onsite measurements, with very reasonable variance.

Conclusion

This paper presents an alternative methodology for solving complex hydraulic problems using 3D numerical modelling. This methodology is used when problems are too complex (because of too many variables) that manual calculations and imperial methods are neither accurate nor sufficiently detailed.

Apart from river flow simulation, this methodology has also been used in many other applications for various hydraulic structures, such as:

- Flow simulation at water intake structures, including sedimentation simulation
- Flow simulation at the powerhouse tailrace
- Flow simulation at the dam sluiceway, for hydraulic performance and energy dissipation
- Flow simulation in the penstock bifurcation or manifold
- Flow simulation through hydraulic mechanical equipment (e.g., valve, turbine, etc.), including erosion simulation

This whitepaper is part of a series of whitepapers for CFD/Ansys applications in various fields.

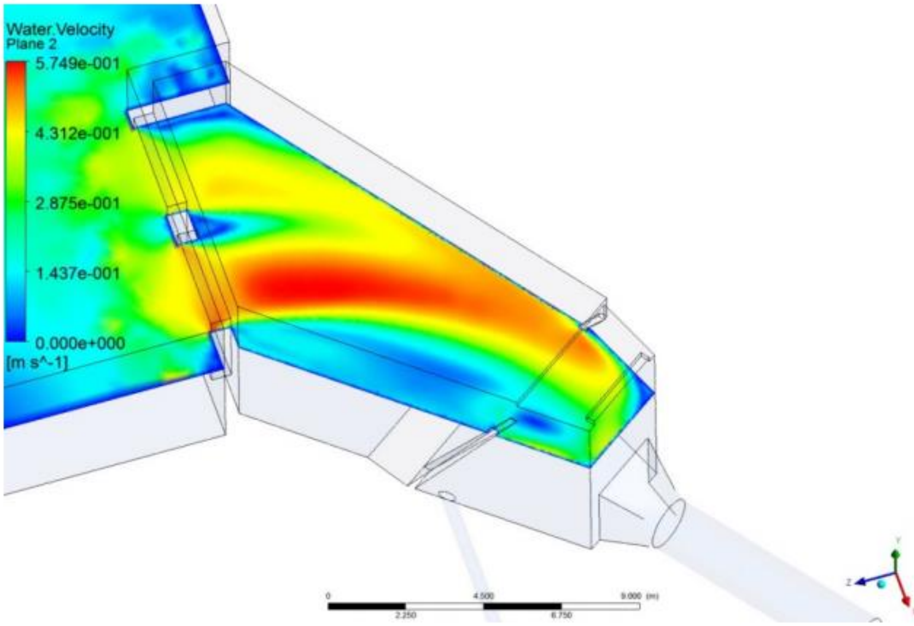

Figure 9: Flow at water intake structure (example)

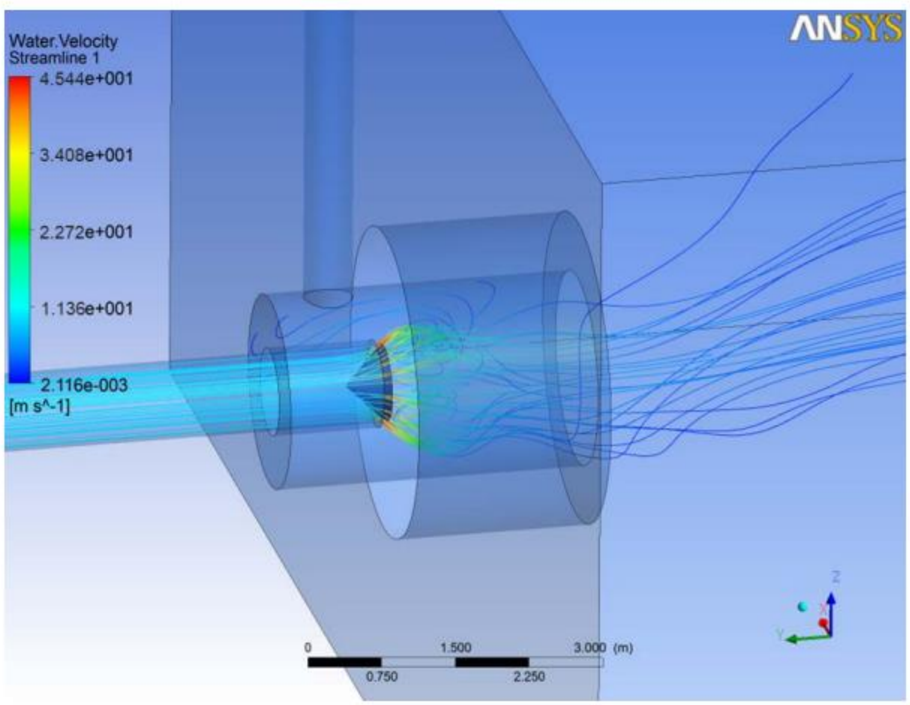

Figure 10: Flow through submerged cone valve at powerhouse tailrace (example)

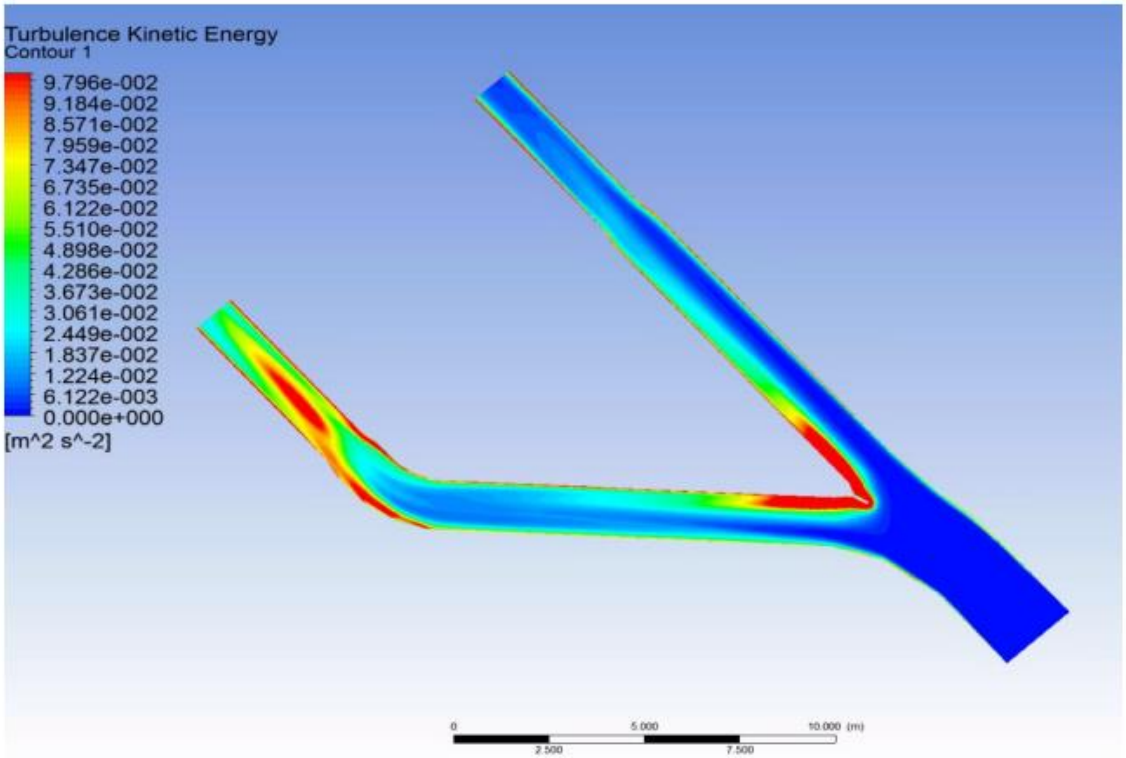

Figure 11: Head loss distribution at penstock bifurcation (example)

About the Author

Glen Ichikawa P.Eng.
Senior Hydropower Engineer
glen.ichikawa@bba.ca

Mr. Ichikawa has over 34 years of experience in hydropower engineering. He has experience in designing tunnels, concrete dams/weirs, spillways, intakes, canals, pipelines, penstocks and powerhouses (both hydraulically and structurally). He has an extensive background in hydrology, energy and power generation modelling along with expertise in the design of buried and surface penstocks ranging in diameters from 16 to 148 inches and heads of over 1000 m. He has supervised the design and construction of projects ranging from 10 kW to over 160 MW. Over the past 14 years, Mr. Ichikawa has worked on 26 clean energy projects that have been built or are under construction.

He has worked closely with EPC contractors in developing successful bids and detailed design engineering services that include construction supervision. He has experience in site investigations, conceptual to final designs, and construction supervision of hydroelectric schemes in North, Central and South America, the Caribbean and Asia. He has managed and coordinated multidisciplinary teams of geologists, surveyors, borehole drillers, seismic geophysicists, hydrologists and geotechnical engineers in the field. He has managed and coordinated the overall design team on a number of projects that involve hydraulic, civil, structural, geotechnical, electrical and mechanical design engineers. In 2014, Glen was honoured with a Lifetime Achievement Award by Clean Energy BC.

BBA.CA