Hybrid modeling and design review of throttled surge tanks

Mona Seyfeddine
Platform of Hydraulic Constructions (PL-LCH)
Ecole Polytechnique Fédérale de Lausanne (EPFL)
EPFL ENAC IIC PL-LCH
Lausanne
Switzerland

Samuel Vorlet
Platform of Hydraulic Constructions (PL-LCH)
Ecole Polytechnique Fédérale de Lausanne (EPFL)
EPFL ENAC IIC PL-LCH
Lausanne
Switzerland

Nicolas Adam
Alpiq SA
Chemin de Mornex 10
Lausanne
Switzerland

Bernard Valluy
Alpiq SA
Chemin de Mornex 10
Lausanne
Switzerland

Christophe Nicolet
Power Vision Engineering Sàrl
Chemin des Champs-Courbes 1
Ecublens
Switzerland

Giovanni De Cesare
Platform of Hydraulic Constructions (PL-LCH)
Ecole Polytechnique Fédérale de Lausanne (EPFL)
EPFL ENAC IIC PL-LCH
Lausanne
Switzerland

Introduction

The Swiss energy strategy 2050 aims to abandon nuclear energy and completely rely on renewable energy production by the year 2050. The goal of this strategy is to increase the energy efficiency and promote the development of renewable energies. The intermittence issue of the new renewables (such as wind or solar energy) raised the need for an increase in hydropower production in order to sustain the grid balance.

Hydroelectric power plants represent around 60% of the annual electricity production in Switzerland and are therefore fundamental to the electricity supply sector [1]. Their production is planned to rise from 35’350 GWh (reference year 2011) to an average annual production of 38’600 GWh in 2050 [1].

The increase in hydropower productions could be done by either constructing new hydropower plants or refurbishing and rehabilitating existing ones. The idea is to make use of the latest advances in technology to have a more efficient usage of the available hydropower potential, a large part of which is attributed to high-head power plants.

The refurbishment of such high-head power plants often requires a modification of the existing surge tanks to handle the new design loads. Throttling the existing surge tanks has been commonly used as a cost-effective solution to adapt hydropower plants subjected to a moderate increase of installed capacity [2]–[6]. There are different types of throttles restricting the entrance to a surge tank and depending on the case, they can either be placed at the entrance of the surge tank (i.e. at the connection between the tank and the pressure tunnel) or in the intermediate shaft between two expansions [3], [5], [7], [8]. These devices offer the advantage of introducing distinct head losses that can keep the extreme mass oscillations within the surge tank, thus eliminating the need to further modify its geometry.

This study is concerned with the numerical modeling of Gondo high-head power plant (HPP) in Switzerland which has been recently subjected to a moderate increase of discharge. In a previous study, a 1D numerical transient analysis identified the need for installing a throttle at the inlet of the existing surge tank. This device introduces asymmetrical head losses, and its geometrical design was optimized by means of physical modeling. The final selected geometry is a rack throttle consisting of a framework of parallel spaced bars. The measured head loss coefficients of this geometry were reincorporated in the 1D numerical model which was later validated thanks to the availability of prototype measurements. Numerical investigations by means of 3D numerical modeling are later conducted and deemed necessary for a better understanding of the performance of the designed throttle. This paper summarizes the methods and the results associated with each modeling strategy, highlighting the importance and effectiveness of adopting a hybrid modeling approach.

1 Gondo high-head powerplant: description and location

Gondo power plant is located in the Canton of Valais in Switzerland. It was commissioned in 1952 and is managed by Energie Electrique du Simplon SA (EES) along with the powerplants of Gabi and Tannuwald. The reservoir supplying Gondo power plant is mainly formed by the Sera arch dam, and Gabi (11 MW) and Tannuwald (5 MW) powerhouse outlets as illustrated in Fig. 1.

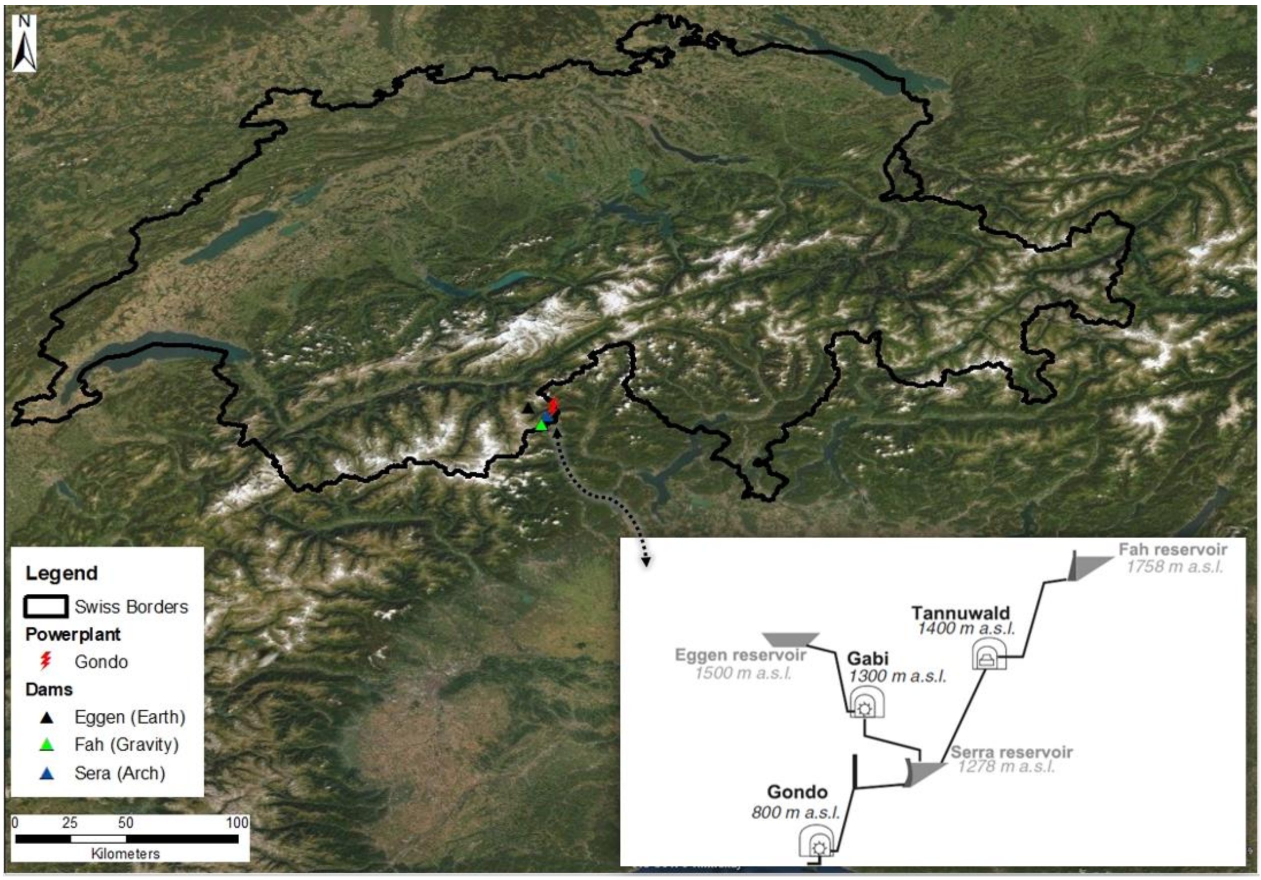


Fig. 1. General location of the Gondo powerplant and a schematic view of the working stage of EES (adapted from [2] using (ArcGIS, 2017))

The pressure tunnel and the first part of the pressure shaft are concrete-lined, while the second lower part of the pressure shaft (starting at an altitude of 1,040 m a.s.l) is steel-lined. An inclined surge tank, consisting of lower and upper expansion chambers connected by an intermediate shaft, is installed at the intersection between the pressure tunnel and the pressure shaft in order to protect the tunnel against transient phenomena and allow fast turbine maneuvers.

The plant was initially operating with two 18.5 MW Pelton turbines exploiting a 470-meters head at a discharge of 11 m^3/s up until the 80's when a third Pelton turbine (8 MW) was installed. The discharge was then increased to 12.1 m^3/s. The goal of the project "Renewal of Group 3" was to replace the third turbine by a more efficient and powerful one, which allows increasing the discharge flowing through the plant up to 14.7 m^3/s.

2 1D Numerical Modeling: Simsen

The increase in the installed generation capacity requires a verification of the ability of the pressure tunnel and shaft to handle the dynamic pressure and maintain the stability of the lining. It also requires a verification of the surge tank extreme levels compared to the required minimum and maximum ones, since an increase in discharge induces an increase in the maximum water level and a decrease in the minimum water level for the same opening or closure time.
This verification was done with a 1D transient model using SIMSEN. This software solves one-dimensional continuity and momentum equations using the finite-difference method in an analogy with electrical schemes. The 1D numerical model was first calibrated with on-site measurements during an emergency closure of the three turbine groups, that took place prior to the refurbishment of the plant. Then, a comprehensive transient analysis focusing on the following normal load cases was done in order to identify potential problems associated with the power increase [2], [4]:

- Emergency shutdown of all units
- Simultaneous loading of all units
- Loading followed by emergency shutdown at the worst moment for the upsurge in the surge tank
- Load rejection followed by a reloading while all units remain connected to the grid
- Emergency shutdown as well as loading and emergency shutdown leading to the closure of injectors in the penstock reflection time (so-called Peak of Michaud)

The analysis showed that the increase in discharge from 12.1 to 14.7 m^3/s could be safely done if the closure time of the injectors is increased to fulfill admissible maximum pressure in the penstock in the Peak of Michaud load case, and if a solution is found to prevent surge tank dewatering in the case of unloading followed by a unit reloading.
Since extending the chambers wasn’t practically feasible, an effective and economical solution to adapt the existing surge tank to a small increase of installed capacity was the placement of a throttle at the bottom of the surge tank. The required head loss coefficients of the throttle in each flow direction, were identified by an iterative optimization process using the 1D numerical simulations. A head loss coefficient of around 30 for inflow and between 40 and 76 for outflow would guarantee an optimal transient response over the entire upper reservoir water level range considering the discharge limitation for the reduced water levels.
To achieve the target head losses determined by the numerical model, the design of the throttle was optimized through an experimental campaign due to the complex geometry of the junction between the pressure tunnel and the surge tank.

3 Physical Modeling

The physical model was constructed at the Platform of Hydraulic Constructions (PL-LCH), Ecole Polytechnique Fédérale de Lausanne (EPFL) with a geometric scale of 1/12. It was done in Froude similarity which despite being less common in pressurized flow than Reynolds similarity and more common in free surface flow, could be used in this case since the lowest Reynolds number *Re* ensures a fully turbulent behavior and head losses do not depend on *Re*. In fact, they are proportional to the kinetic energy of the flow.

The boundaries of the physical model are limited to the inclined surge tank and the confined pressure tunnel and pressure shaft stretches (Fig. 2).

Several alternatives of the throttle’s geometry (gate and rack throttles) were tested, and for each geometry four different flow directions are investigated: generation turbining flow (C-B), turbine startup (A-B), flow going into the surge tank during mass oscillation (C-A) and finally the level decrease of mass oscillation (A-C).

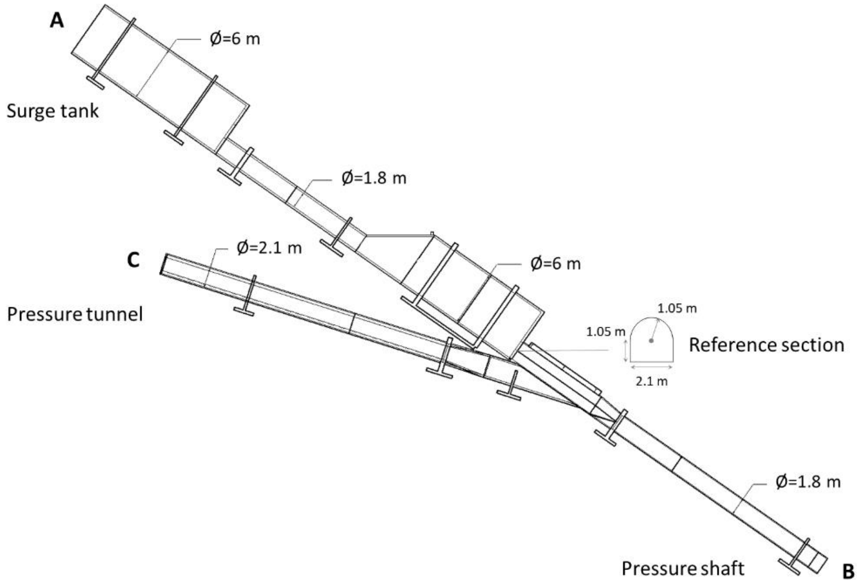


Fig. 2. Experimental set-up; reference section at the bottom of the surge tank (prototype dimensions)

The head losses are evaluated using steady-state tests for 5 different discharges. The tests start with the maximal flow which is gradually reduced till the minimal flow is reached. Then, the flow is gradually re-increased in order to reach the maximal flow. This results in two measurements for each flow, improving therefore the accuracy and reliability of the analysis.

Depending on the investigated flow direction, the head loss is evaluated between two control sections S_i and S_j according to the Bernoulli equation:

$$\Delta H_{ij} = E_i - E_j = (z_i + \frac{P_i}{\gamma} + \frac{v^2{}_i}{2g}) - (z_j + \frac{P_j}{\gamma} + \frac{v^2{}_j}{2g}) \quad (1)$$

By considering negligible linear losses in the system, this head loss mainly consists of singular steady-state losses which could be expressed as a linear function of the kinetic energy:

$$\Delta H = k\ \frac{v^2{}_{ref}}{2g} \qquad (2)$$

V_{ref} being the velocity in the reference cross-section based on which k is evaluated. This section is located at the intersection between the surge tank bottom chamber and the junction between the pressure tunnel and the pressure shaft (Fig. 2).

The final selected geometry is a rack throttle consisting of parallel spaced bars (trapezoidal beams), it is best illustrated in Fig. 3.

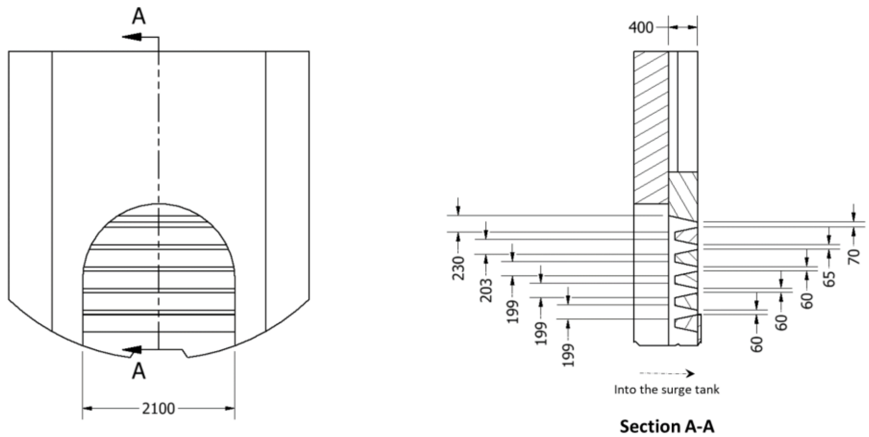


Fig. 3. Front (left) and section view (right) of the rack throttle (prototype dimensions, millimeters)

The head loss coefficients of this optimal geometry were computed with the least square method and with respect to the reference section located at the bottom of the surge tank (Fig. 2). They are presented below for each flow direction.

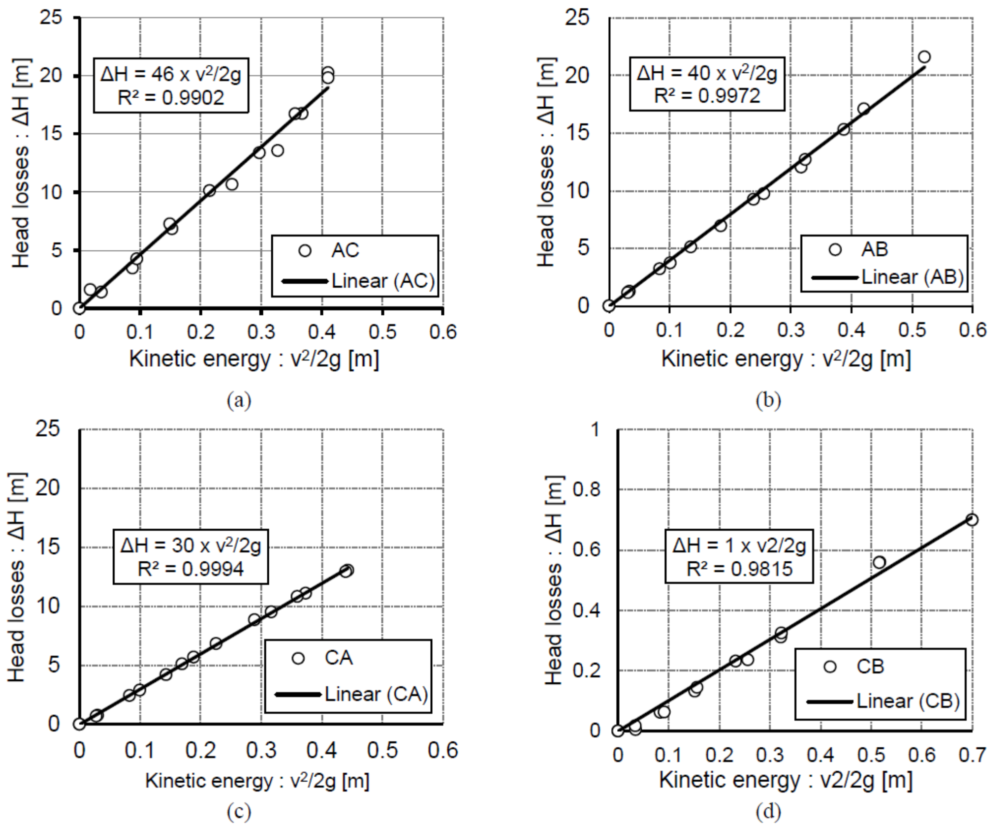


Fig. 4. Head loss coefficients obtained with the least square method for each flow direction: (a) Flow going out of the surge tank during mass oscillation, (b) flow going out of the surge tank during turbine opening, (c) flow into the surge tank during mass oscillation and (d) steady turbining flow [4]

4 Validation of the 1D numerical model

The head loss coefficients of the throttle measured in the physical model were reimplemented in a refined Simsen 1D numerical model. To ensure that these two models reflected the real behavior of the system, prototype measurements (following an emergency shutdown of turbine groups G1 and G2) were done several months following the installation of the rack throttle. Note that at the period of measurement, the new 3rd turbine group was not installed, so the valve on the axis of group 3 was initially shut. Numerical results showed good agreement with the on-site measurements as presented in Fig. 5 below.

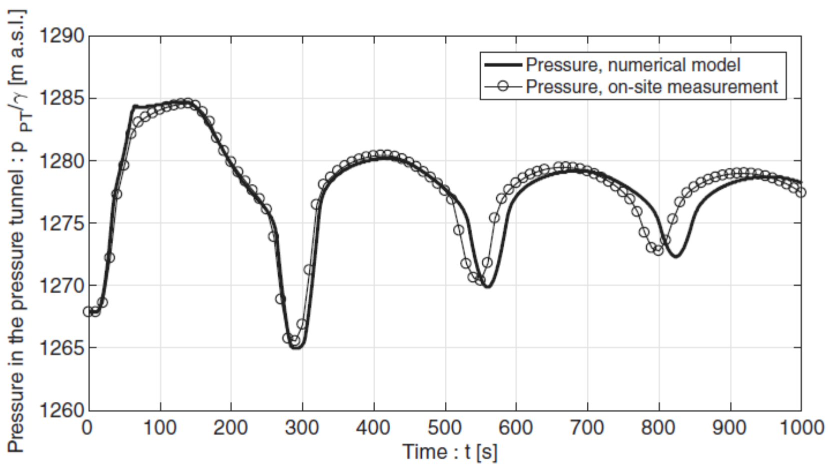


Fig. 5. Comparison between on numerical and in-situ measurements following an emergency shutdown of G1 and G2 units; (upper reservoir at 1,277.6 m a.s.l. (April, 2015) [2]

In the context of hybrid modeling, and as a way to extend the previous analyses conducted on Gondo high-head power plant, it has been found necessary to construct a 3D numerical model. Unlike physical scale modeling, the 3D numerical model allows the observation of flow patterns at any desired location in the system constructed at the prototype scale. The latter also serves to validate the experimental values of the throttle's head loss coefficients and offers the opportunity to assess factors that 1D numerical modeling or physical scale modeling fail to accurately depict. Such findings could be beneficial for future designs of throttled surge tanks.

5 Numerical 3D modeling: ANSYS CFX

The chosen numerical code for the 3D analysis is ANSYS CFX (version 2019 R1). Below is a summary of the background behind this code, the key simulation steps and the main results of the 3D simulations.

5.1 Background

Computational Fluid Dynamics (CFD) is a computerized tool that allows simulating the behavior of fluid flow systems among many other physical processes, by solving the basic equations of fluid flow over a region of interest with specified boundary conditions [9]. The set of equations dictating the physics of fluid mechanics is known as the Navier-Stokes equations. They are based on the three conservation laws of physics (mass, momentum, and energy) and have no known general analytical solution. Nevertheless, they can be discretized and solved numerically through a variety of solution methods the most common of which is the finite volume method [9]. ANSYS CFX is based on this technique, which divides the region of interest into a set of control volumes (small sub-regions) on which continuous partial differential equations of conservation are discretized into a system of linear algebraic equations and solved iteratively [9].

5.2 Simulation Steps

5.2.1 Geometry

The geometrical boundaries to the CFD model are chosen similarly to the physical model (Fig. 6). The overall system consisting of the surge tank, pressure tunnel, pressure shaft and the diaphragm, is sketched in prototype (real) dimensions.

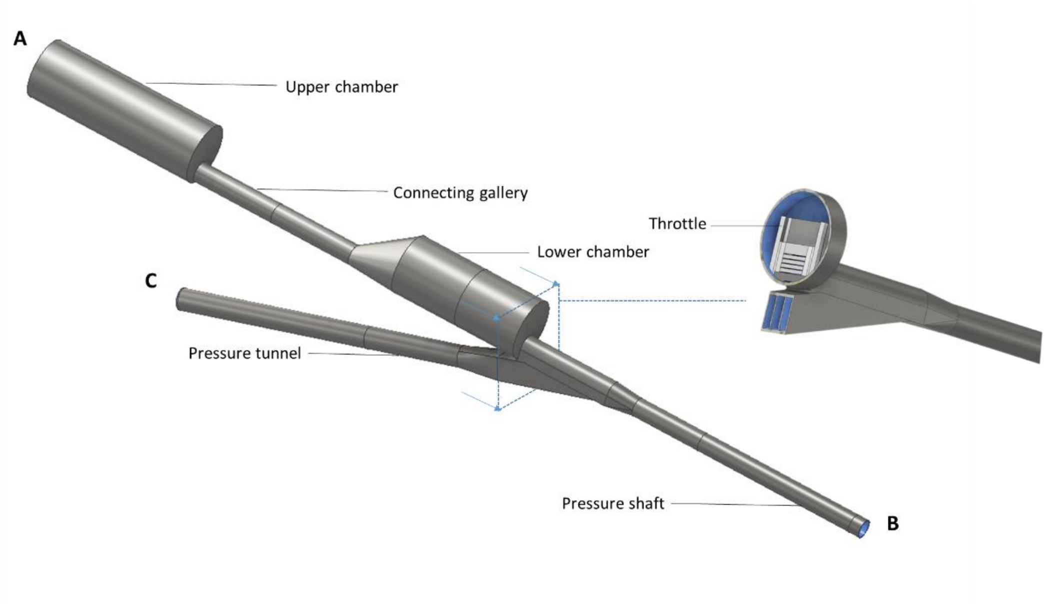


Fig. 6. Boundaries of the 3D numerical model; section view at the bottom of the surge tank

5.2.2 Meshing

The meshing process should be an optimization between the degree of accuracy (mesh precision) and the computation (compilation) time. The fluid volume was meshed with tetrahedral elements under the patch conforming algorithm. This technique guarantees that the faces of the fluid body along with their boundaries are respected within a very small tolerance [10].

Mesh refinements were done in regions of interest, i.e. sharp edges, bends, throttle elements and the connecting gallery of the surge tank. Additionally, edge sizing control was applied to the spacings between the bars of the throttle. Several element sizes were tested and the size that ensured a grid-independent solution with the least number of elements was determined with a mesh-sensitivity analysis presented in *section 5.3.*

Fig. 7 illustrates the following: a global view of the mesh (unstructured), edge sizing around the throttle and a local refinement around the connecting gallery of the surge tank in order to accurately predict the minor losses caused by the variation in the section of the surge tank.

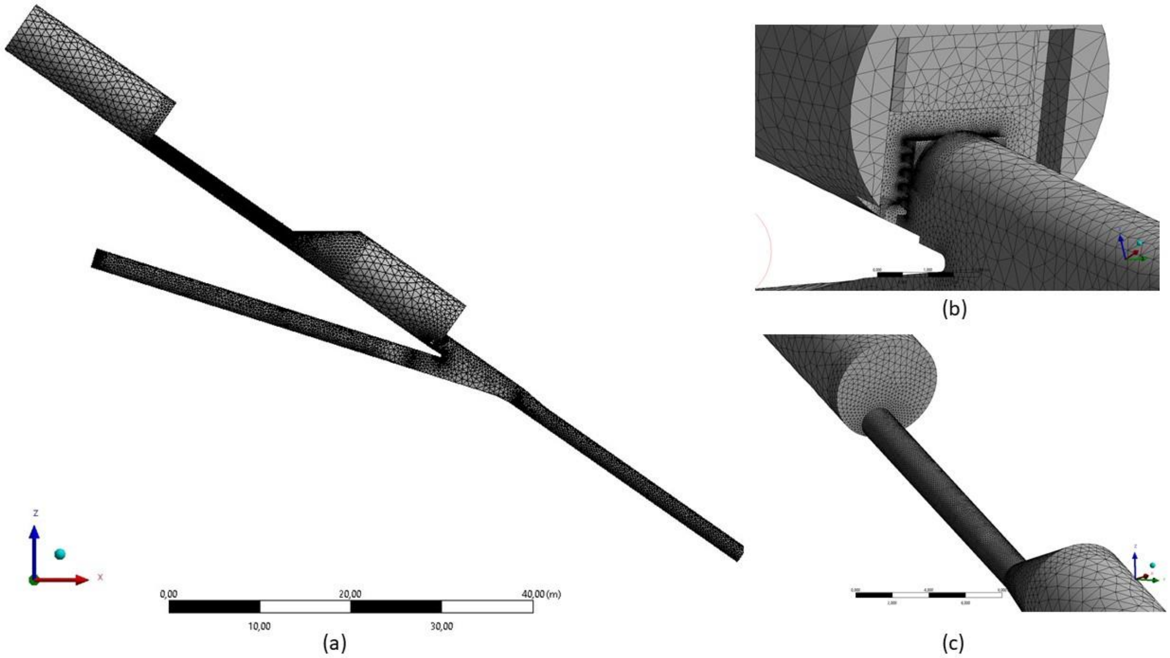


Fig. 7. Mesh illustration: (a) Global view of the mesh; (b) edge sizing around the throttle; (c) local refinement around the connecting gallery of the surge tank (ANSYS CFX)

5.2.3 Pre-processing

Mesh files are loaded into the physics pre-processor CFX-Pre, where the fluid properties and boundary conditions are specified.

Physical parameters: The fluid consists of water assessed at a temperature of 15 degrees. Steady-state analyses are sufficient to predict the head loss coefficient corresponding to a certain flow direction. Hence, the surge tank is always full, and the fluid is in a single phase. However, for investigations aiming to predict water level variations in the surge tank taking place under transient conditions, a multi-phase flow should be used to correctly depict the air-water interaction.
As the flow in the system is gravity driven, buoyancy was activated, and the gravity was set in the negative Z-direction (-9.81 m/s^2).

Boundary conditions: Depending on the investigated scenario, each boundary is either set as inlet, outlet or opening. Uniform pressure and velocity profiles are entered as boundary conditions for the inlets and outlets. 5 different discharges are tested for each investigated flow direction.
Regarding the pipe and surge tank walls, a no-slip boundary condition is set in order to ensure a zero fluid velocity immediately next to the walls. A roughness was applied to the wall surfaces and an equivalent sand-grain roughness of 0.27 mm is used as an input parameter.

Turbulence model: the Shear Stress Transport (SST) model is used to provide closure for Reynolds-averaged Navier-Stokes equations. SST is a combination between the k–ω and the k–ε models. SST uses a transformation of the k–ε into a k–ω in the near-wall region, and the standard k–ε model in fully turbulent regions that are far from the wall [11].
This model has been proven to yield sufficient accuracy in predicting onset and amount of flow separation, as validated by comparable investigations in the literature [3], [8], [12].

5.2.4 Solver

The solver is the component that solves the CFD problem by integrating the partial differential equations over all the control volumes in the region of interest. These equations are converted to a system of algebraic equations which are then solved iteratively.

As a measure of the accuracy of the solution, two convergence criteria are set in the solver control settings:

- RMS residual, based on the average residual from all control volumes, set at 1.0E-4
- Conservation target, which sets a target for the global imbalances as a measure of the overall conservation of mass, momentum and energy in the flow domain, set at 1.0E-2

5.2.5 Post-processing

Post-processing allows the visualization and analysis of the results.

The value of the throttle's head loss coefficient *k* in each flow direction, is obtained using the least square method based on the reference cross section located at the bottom of the surge tank. The reference sections for the calculation of the head loss are chosen in zones experiencing uniform flow conditions, at the same locations of the ones chosen with the physical model in order to have the same basis for comparing the values of *k* as obtained with each method.

5.3 Mesh-sensitivity analysis

To ensure a grid-independent solution for predicting the head losses across the throttle, a mesh sensitivity analysis was conducted with A-B as a reference flow direction. Five types of mesh around the spacings between the bars were tested: one with no edge sizing, and 4 meshes with an edge size of 10, 6, 5 and 4 mm respectively.

By applying the same boundary conditions (constant pressure at the inlet and a constant velocity at the outlet) on the different meshes, the head loss between the inlet (boundary A) and the outlet (boundary B) was monitored for several flows.

The results presented below show that the mathematical relationship between the head loss and the discharge varies with the different mesh sizes. A mesh-independent solution is reached at a maximal edge size of 5 mm.

The smallest spacing between the bars is around 60 mm (kindly refer to Fig. 3); this means that a cell size representing at most 1/12 of the actual spacing size was proven to yield sufficient accuracy in the head loss prediction.

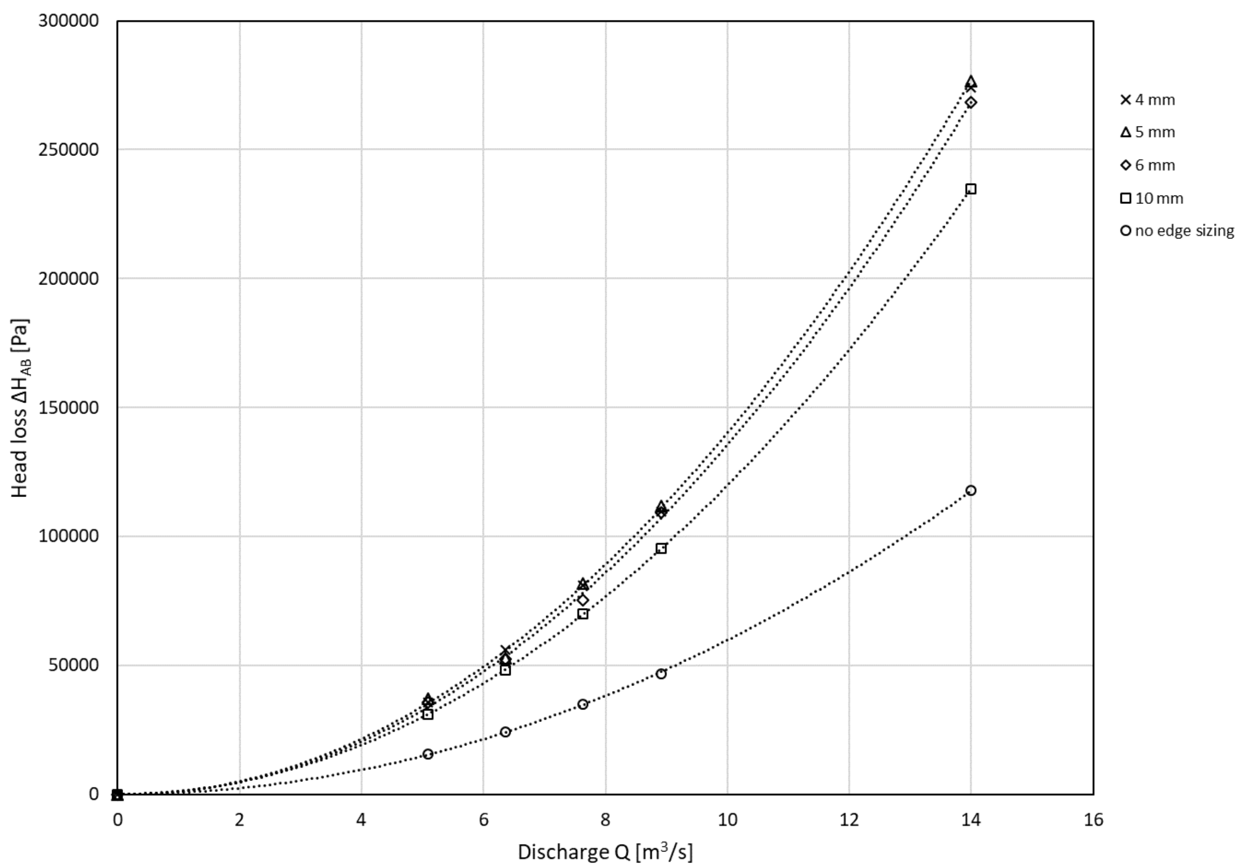


Fig. 8. Head loss between inlet A and outlet B as a function of discharge for 5 different meshes

Fig. 9 underlines the reason behind the differences in the obtained flow characteristics between the different meshes. Under the same flow conditions (Q=14 m^3/s), a mesh with no edge sizing (a) completely fails to capture the contraction of the streamlines through the spacings of the throttle, resulting in an underestimated velocity profile around the throttle when compared to the refined mesh with a maximal edge sizing of 5 mm (b).

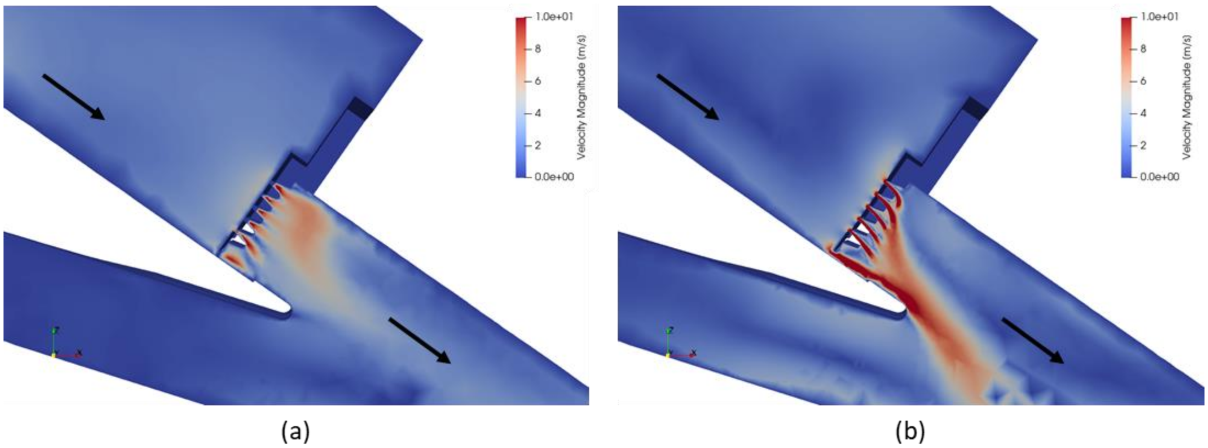


Fig. 9. Velocity contour in the vicinity of the throttle, for Q=14 m^3/s: (a) no edge sizing, (b) maximal edge sizing of 5mm

Insufficient refinement can highly underestimate the head losses throughout the throttle, reaching a 60% underestimation of *k* for the case of no edge sizing compared to the value obtained using 5 mm and 4 mm edge sizing.

5.4 Results

5.4.1 Head loss coefficients and flow patterns

The head loss coefficients for the four flow directions are obtained by linear regression (with respect to the same reference section as the physical model). Results along with illustrations of the velocity streamlines are presented for all the directions under the same flow conditions to allow for comparison with one another. These illustrations help in clarifying the differences in the numerically computed values of *k*.

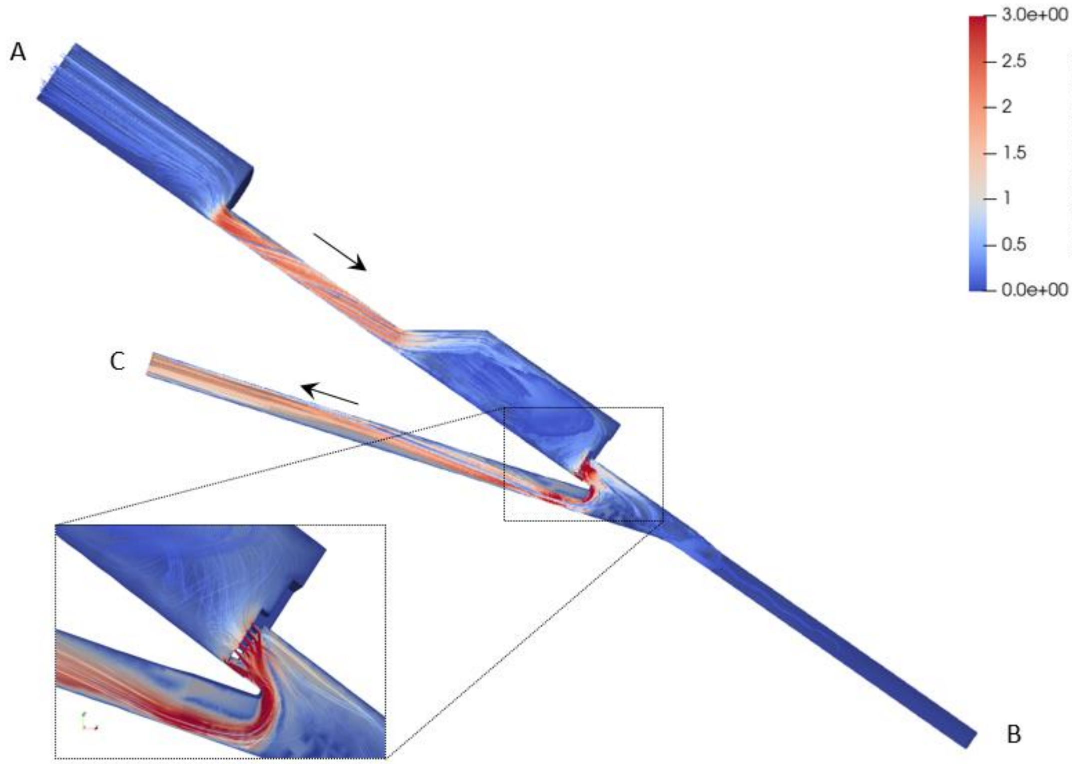


Fig. 10. Velocity streamlines for a 5 m^3/s flow going from A to C ($\mathbf{k_{AC} = 45.2}$)

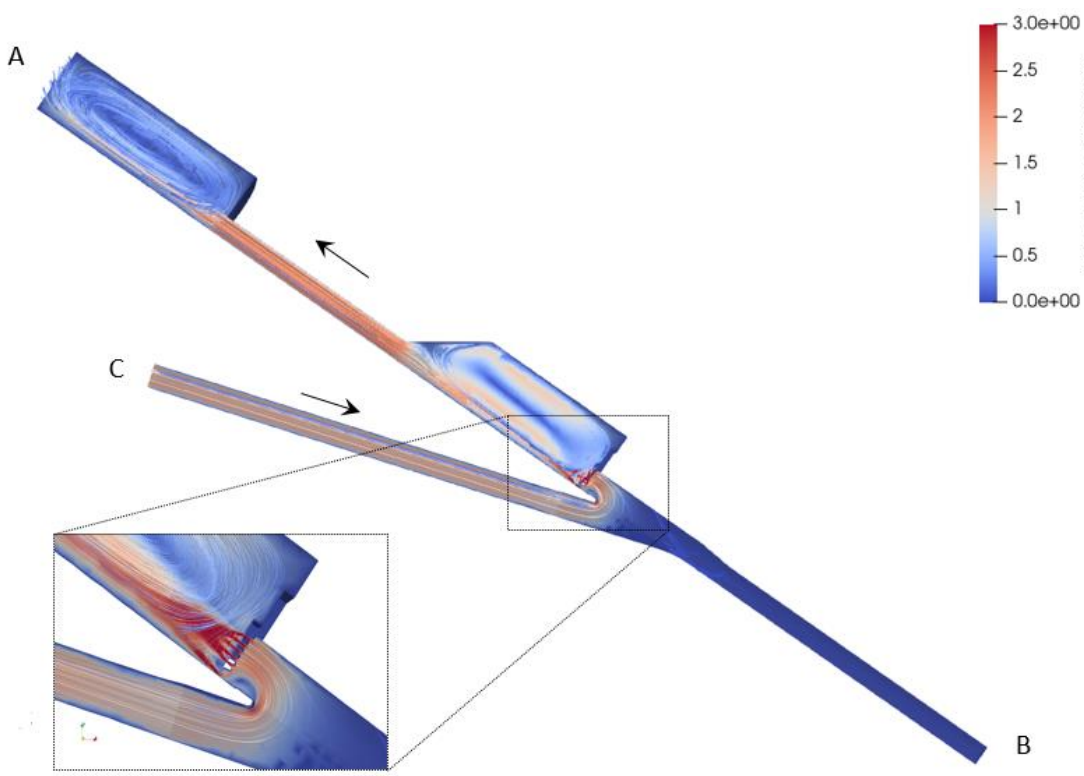


Fig. 11. Velocity streamlines for a 5 m^3/s flow going from C to A ($\mathbf{k_{CA} = 28.2}$)

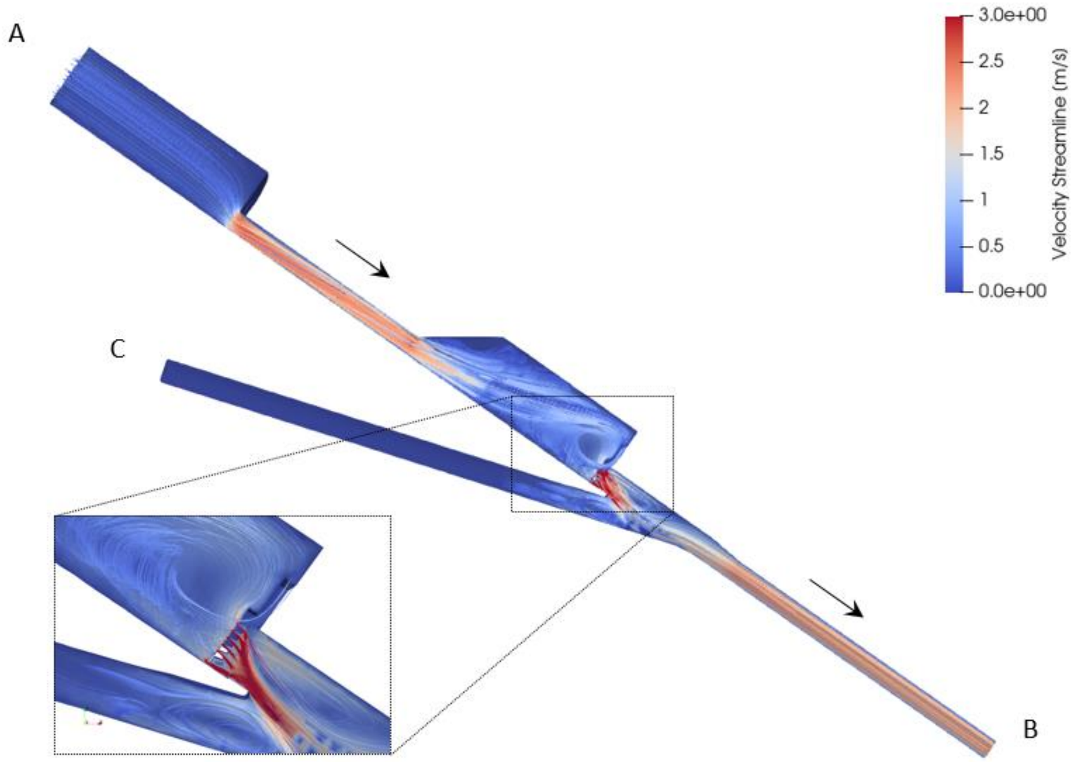


*Fig. 12. Velocity streamlines for a 5 m³/s flow going from A to B (**k_{AB} = 42.1**)*

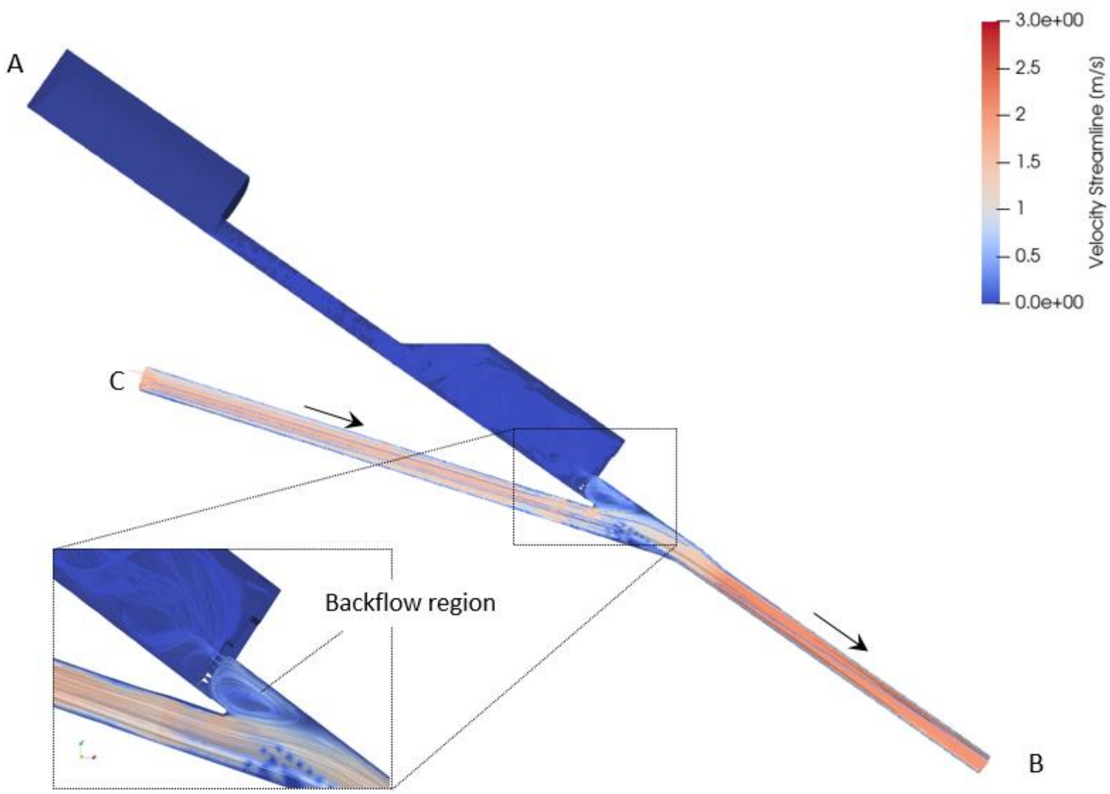


*Fig. 13. Velocity streamlines for a 5 m³/s flow going from C to B (**k_{CB} = 0.86**)*

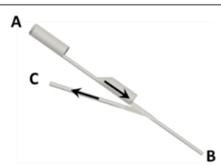
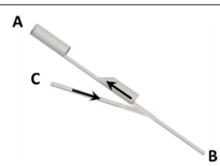
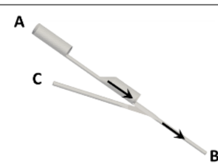
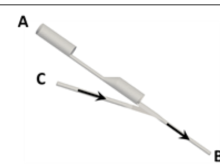
By taking a close look at the throttle in Fig. 10, Fig. 11 and Fig. 12, it can be seen that the lower chamber of the surge tank experiences high flow disturbances accompanied by recirculation zones. For the same flow of 5 m^3/s, streamlines exiting the surge tank (A-C, A-B) experience a more chaotic or disturbed behaviour than the ones entering the surge tank (C to A). The orientation of the trapezoidal bars plays a crucial role in guiding the flow into the surge tank, and the more guided it is, the less are the head losses. This results in higher head loss coefficients for water going out of the surge tank (A-C or even A-B) than water going into the surge tank (C-A), and therefore satisfies the asymmetry requirements of the throttle as identified by the 1D numerical transient study.

Regarding Fig. 13, for the flow going in the C-B direction (steady flow during turbine generation), the flow undergoes shearing in the junction between the pressure tunnel, shaft and surge tank. The latter results in a backflow region in the vicinity of the throttle. Since the streamlines do not pass through the throttle, the local head losses are limited to the bend and the backflow region, which justifies the low value of *k*.

5.4.2 Comparison with the physical model results

The following table summarizes the head loss coefficients for each flow direction as obtained with the physical model and with the ANSYS CFX 3D numerical model. The relative difference is computed with respect to the experimental value.

Table 1. Throttle's head loss coefficients (numerical and experimental values) for each flow direction

FLOW DIRECTION				
EXPERIMENTAL VALUE [-]	k_{AC} = 45.9	k_{CA} = 29.6	k_{AB} = 39.8	k_{CB} = 0.98
NUMERICAL VALUE [-]	k_{AC} =*45.2*	k_{CA} = 28.2	k_{AB} = 42.1	k_{CB} = 0.86
RELATIVE DIFFERENCE [%]	*1.5*	*4.7*	*5.8*	*12.2*

The CFD model results successfully line up with the physical model in terms of head loss characteristics for flow shifting directions from a side to another. The head loss coefficients through the throttle for all flow directions were found in excellent agreement with the ones of the scale model (maximum relative difference of 6%).

Generally, Darcy-Weisbach type friction losses are not modelled accurately in CFD 3D simulations. The weakness of the turbulence models and the difficulty of the boundary mesh to solve small-scale roughness elements, typically result in underestimated friction factors when compared to measured experimental values [13]. This does not seem to significantly affect the loss coefficients in directions AC, CA, and AB in which the head losses are dominated by the local ones induced by the throttle. However, in the CB direction in which the throttle is not involved, the head losses are merely attributed to the frictional Darcy-Weisbach losses as well as the minor losses due to the bend and the backflow region formed on top of it. The low numerical value of *k* was therefore found to be smaller than the experimental one, resulting in a higher percent difference than the previous cases.

The good agreement achieved between the two methods confirmed the need for using the 3D numerical code ANSYS CFX as a reliable method to predict local head loss coefficients for throttled surge tanks. Several studies conducted in this connection provided further confidence [3], [12].
As the physical model values were implemented in the refined 1D numerical model, which was later validated with on-site measurements, discrepancies are believed to stem from the following errors or uncertainties in CFD modeling [11]:

- errors in the simulations: rounding errors or convergence criteria that are not strict enough, discretization errors due to inadequate mesh
- uncertainty in the CFD model: no CAD model is illustrative of the exact geometry constructed on a reduced or a prototype scale. Additionally, uniform conditions (pressure, velocity) on surface boundaries and the chosen turbulence model are approximations that may not always depict the complexities of the real flow conditions

Sufficient attention should be given to the issues cited above and a mesh sensitivity analysis is crucial to make sure that the results are grid-independent. As highlighted in *section 5.3*, an insufficient mesh accuracy may fail in capturing the contraction of the streamlines in narrowed spacings which can result in misleading conclusions.
Validation with physical modeling is an important step to overcome all the possible limitations accompanying 3D numerical models. Ultimately, the geometry optimization of an asymmetrical throttle in a complex system, may be done using preliminary CFD models before being tested and validated on a physical scale. This could save time and cost by reducing the number of likely iterations that should be done on a physical scale and might help identify any critical aspects of the design that should not be overlooked.

6 Conclusion and outlook

This paper focused on Gondo HPP that has been recently refurbished by a moderate increase of installed capacity. The refurbishment required an adaptation of the existing surge tank by mounting a throttle at its entrance. The goal of the current study was to extend previous 1D numerical and physical model analyses by a 3D numerical model in addition to highlighting the importance of adopting a hybrid modeling approach combining all means of modeling to ensure an optimum solution.
The CFD model, developed using ANSYS CFX computational fluid dynamics tool, has the advantage of modeling the system in real dimensions in addition to visualizing flow patterns in internal sections, which were difficult to examine in the scaled physical model. It confirms the findings of the physical model regarding the head loss coefficients of the throttle. The conducted investigations provide an insight on the real behavior of a hydraulic system equipped with a throttle and are likely to improve the future design of throttled surge tanks. Additionally, they enhance the confidence in 3D numerical modeling which can provide preliminary convenient conclusions prior to physical modeling, decreasing therefore the number of likely costly modifications with the latter.

7 Acknowledgements

The authors would like to express their gratitude to the company Etaeval who performed measurements during the first transient test campaign for model calibration, and also to the staff of Hydro Exploitation SA involved in the power plant operation and supervision during transient tests and performed the second transient test campaign as well as the measurements of the detailed 3D geometry of the surge tank.

References

[1] **Office fédéral de l'énergie OFEN**, "Potentiel hydroélectrique de la Suisse : Évaluation du potentiel de développement de la force hydraulique dans le cadre de la Stratégie énergétique 2050," Département fédéral de l'environnement, des transports, de l'énergie et de la communication DETEC, Août 2019.

[2] **N. Adam, G. De Cesare, C. Nicolet, P. Billeter, A. Angermayr, B. Valluy, and A. J. Schleiss**, "Design of a Throttled Surge Tank for Refurbishment by Increase of Installed Capacity at a High-Head Power Plant," *J. Hydraul. Eng.*, vol. 144, no. 2, p. 05017004, Feb. 2018, doi: 10.1061/(ASCE)HY.1943-7900.0001404.

[3] **S. Alligne, P. Rodic, J. Arpe, J. Mlacnik, and C. Nicolet**, "Determination of Surge Tank Diaphragm Head Losses by CFD Simulations," in *Advances in Hydroinformatics*, P. Gourbesville, J. Cunge, and G. Caignaert, Eds. Singapore: Springer Singapore, 2014, pp. 325–336.

[4] **G. De Cesare, N. J. Adam, C. Nicolet, A. Angermayr, and B. Valluy**, "Surge tank geometry modification for power increase," in *Hydro 2015 : Advancing policy and practice*, Bordeaux, France, Oct. 2015, p. 16.

[5] **F. Hachem, C. Nicolet, R. Duarte, G. De Cesare, and G. Micoulet**, "Hydraulic Design of the Diaphragm's Orifice at the Entrance of the Surge Shaft of FMHL Pumped-storage Power Plant," p. 10, 2013.

[6] **Y. T. Li**, “Orifice head loss in the T-section of a throttled surge tank,” *Water Power*, vol. 24, no. 9, pp. 326–334, 1972.
[7] **N. Adam, G. De Cesare, and A. Schleiss**, “Surge tank throttles for safe and flexible operation of storage plants,” p. 8, 2016.
[8] **W. Richter, W. Dobler, and H. Knoblauch**, “Hydraulic and numerical modelling of an asymmetric orifice within a surge tank,” in *4th IAHR International Symposium on Hydraulic Structures*, Porto, Portugal, 2012, p. 7.
[9] **ANSYS, Inc.**, “ANSYS CFX Introduction.” ANSYSC, Inc., Nov. 2013.
[10] **ANSYS, Inc.**, “Meshing User’s Guide.” ANSYS, Inc., Nov. 2010.
[11] **H. K. Versteeg and W. Malalasekera**, *An introduction to computational fluid dynamics: the finite volume method*, 2nd ed. Harlow, England ; New York: Pearson Education Ltd, 2007.
[12] **R. Gabl and M. Righetti**, “Design criteria for a type of asymmetric orifice in a surge tank using CFD,” *Eng. Appl. Comput. Fluid Mech.*, vol. 12, no. 1, pp. 397–410, Jan. 2018, doi: 10.1080/19942060.2018.1443837.
[13] **R. T. Christensen, R. E. Spall, and S. L. Barfuss**, “Application of Three RANS Turbulence Models to Aged Water Transmission Pipes,” *J. Hydraul. Eng.*, vol. 137, no. 1, pp. 135–139, Jan. 2011, doi: 10.1061/(ASCE)HY.1943-7900.0000288.

The Authors

Mona Seyfeddine graduated with a master’s degree in Civil Engineering at Ecole Polytechnique Fédérale de Lausanne (EPFL) in Switzerland, with a specialisation in Hydraulics and Energy. This publication summarizes the objectives of her master’s thesis that took place at the Platform of Hydraulic Constructions of EPFL.

Samuel Vorlet is currently a scientist at Platform of Hydraulic constructions (formerly Laboratory of Hydraulic Constructions LCH) of École Polytechnique Fédérale de Lausanne (EPFL). He graduated in Civil Engineering from EPFL in 2018. Since then, he works mainly on projects regarding numerical simulation and physical modeling of dam and hydraulic structures. He is a co-opted member of the Swiss Committee on Dams (SCOD) workgroup on “Reservoir Flushing and Emptying”.

Nicolas J. Adam graduated in Civil Engineering from the University of Liège, Belgium, in 2013. He worked for the Laboratory of Hydraulic Constructions for 4 years and obtained his PhD on the topic of hydraulic behavior of orifices in conduits in 2017 from the EPFL. He joined the Hydro Power Generation of Alpiq in 2018 and is working as civil engineer for the dam safety and the flow forecasting for asset optimization.

Bernard Valluy graduated in Civil Engineering from EPFL in 1996 and holds an EMBA from HEC Lausanne since 2008.
He started his career at an engineering consulting firm in Paris as a dam and hydropower engineer before joining EOS now Alpiq in 2003 to work on the Cleuson-Dixence pressure shaft relining project.
He has been involved in new construction of hydropower projects as well as refurbishments in Switzerland and France.
He is currently Head of Project Management and Innovation at Alpiq’s Hydropower Unit where he focuses mainly on Hydropower Asset Development and Digitalization.

Christophe Nicolet graduated from the Ecole Polytechnique Fédérale de Lausanne, EPFL, in Switzerland, and received his master’s degree in mechanical engineering in 2001. He obtained his PhD in 2007 from the same institution in the Laboratory for Hydraulic Machines. Since then, he is the managing director and principal consultant of Power Vision Engineering Sàrl in Ecublens, Switzerland, a company active in the field of optimization of hydropower transients and operation. He is also an external lecturer at EPFL in the field of “Transient Flow”.

Giovanni De Cesare is the operational director of the Platform of Hydraulic Constructions PL-LCH (formerly Laboratory of Hydraulic Constructions LCH) of the Ecole Polytechnique Fédérale de Lausanne EPFL in Lausanne, Switzerland. He is in charge of technology transfer of the hydraulic laboratory, as well as research and teaching. He has more than 25 years of experience in physical and numerical modelling in all domains of hydraulic structures and schemes with more than 100 studies performed as well as several commissioned studies and expert reports in applied hydraulics and hydraulic constructions. He specialises in various fields such as reservoir sediment management and river training work. Giovanni De Cesare holds an MSc in Civil Engineering and a PhD in technical sciences from the EPFL where he studied reservoir sedimentation by turbidity currents with on-site measurements, numerical simulation as well as experimental tests. He is author of more than 150 scientific papers in peer-reviewed journals, publications and conference proceedings.