Conference on Modelling Fluid Flow (CMFF'22)
The 18th International Conference on Fluid Flow Technologies
Budapest, Hungary, August 30-September 2, 2022

FISH INJURY ASSESSMENT OF A HYDROPOWER FACILITY BYPASS

Dennis POWALLA[1], Rishav SAHA[2], Stefan HOERNER[2], Dominique THÉVENIN[2]

[1] Corresponding Author: Laboratory of Fluid Dynamics and Technical Flows, University "Otto von Guericke" of Magdeburg, Universitätsplatz 2, Magdeburg, 39106, Germany, E-mail: dennis.powalla@ovgu.de
[2] Laboratory of Fluid Dynamics and Technical Flows, University "Otto von Guericke" of Magdeburg, Universitätsplatz 2, Magdeburg, 39106, Germany

ABSTRACT

The Las Rives hydropower plant is located on the Ariège river in France. A fish ladder and bypass system was installed in the facility in order to ensure river continuity. The downstream fish pass is located after a rack in front of the intake channel of the power plant. It later connects through a free-stream jet to the tailwater. Objective of this work is to investigate the injury risk for the fish passing the device, caused by possible impacts when penetrating the water surface or hitting the river bed. The study is based on a custom numerical framework combining fluid dynamics with particle-based fish surrogates. In a first step the free stream bypass is modelled with Computational Fluid Dynamics (CFD). The water jet is simulated as an Eulerian multiphase flow, which is described by the Volume of Fluid method (VOF). After establishing a suitable fluid dynamical model of the bypass installation, particles are injected at the bypass inlet while coupling CFD with the Discrete Element Method (DEM). The particles act as fish surrogates in the model. The DEM is a Lagrangian method that allows for tracking contacts between particles and with the wall. Based on the information gained from the unsteady simulations and a multitude of particle tracks, a statistical evaluation of the data regarding fish compatibility is possible. The work presented here is ongoing and provides a first base for further investigations of the site. In a parallel experiment live-fish and passive sensor probes were injected in the bypass by research partners from France and Estonia. The sensors allow for a recording of acceleration and pressure data during the passage. Future work will compare this experimental data and the data gained by the simulations to validate the numerical methodology.

Keywords: Computational fluid dynamics (CFD), Ecohydraulics, Eulerian Multiphase, Hydropower, Reynolds-Averaged Navier-Stokes (RANS), Volume of Fluid (VOF)

NOMENCLATURE

CFD	[]	Computational Fluid Dynamics
CFL	[]	Courant number
DEM	[]	Discrete Element Method
LES	[]	Large-Eddy Simula- tions
RANS	[]	Reynolds-Averaged Navier-Stokes
SIMPLE	[]	Semi-implicit Method for Pressure Linked Equations
SST	[]	Shear stress transport
VOF	[]	Volume of Fluid
k	$[m^2/s^2]$	Turbulent kinetic energy
y^+	$[-]$	Dimensionless wall distance
ω	$[1/s]$	Specific rate of dissipation

1. INTRODUCTION

River continuity is an important aspect of the ecology of water bodies, since it is a requirement for reproduction and migration of a large part of aquatic fauna. Anthropogenic barriers often strongly impact the surrounding ecosystems, in particular for devices that block the natural river continuity. This is typically the case for run-of-river power plants, where a dam interrupts fish passage in both directions, compromising the sustainability of fish populations [1]. In Europe, run-of-river power plant operators have to conform to the European Water framework directive, which states the requirements related to their ecological impact [2]. To enable fish the passage of those obstructions, the current state of the art consists of fish ladders or lifts, which offer alternative paths for fish and reconnect the upper and lower parts of the river otherwise separated by the dam.

It is mandatory to provide information about the mortality and injury risk potential on migrating fish passing either the power plant itself or the upstream and downstream fish passage installations. Especially passing the power plant leads to a higher risk of injury/mortality by collision with the turbine structure or, if a certain threshold of turbine head is exceeded, through barotrauma as shown by Trumbo et

al., 2014 [3] or Bevelhimer et al., 2017 [4]. Up to now these assessments are mostly gained using live fish tests. Giesecke et al. 2014 [5] considers life fish tests to be morally questionable, time-consuming, complex and costly, and in most cases only valid for the specific installation and operating point tested. Considering the limitations and uncertainties of this approach, as well as the very stressful and harmful experience imposed on the animals, an urgent need for the development of alternative prediction models for their replacement can be stated. Already introduced alternative prediction models are so called blade strike models, which can be either of physical or empirical nature. Common physical models are introduced by von Raben ([6]), Monten ([7]) and Turnpenny ([8]) which are based on the same principle and differ in the degree of complexity of the underlying assumptions for the probability of collision between fish and turbine blade. Empirical models such as Larinier and Travade 2002 [9], or Ebel 2008 [10] use statistical evaluation methods by combining observed mortality rates and technical parameters. The physical model have in common that the fish's "behaviour" is simplified as a passive buoyant neutral object carried with the flow, whereas the emperical models require extensive live fish. To address these simplifications and reduce live animal tests further improvements should be developed.

The development of such alternative prediction models is also the aim of the RETERO-project (www.retero.org). This project strives for a reduction of live fish testing for the assessment of any hydropower facility and other descent corridors in installations. This is done by reducing (and in the longer term fully replacing) animal probands by complementary methods such as passive probes, partly-autonomous sensors and numerical simulations. The numerical work presented here employs a combination of Computational Fluid Dynamics (CFD) and Discrete Element Method (DEM) - overlaid by fish behaviour models. CFD-DEM coupling means, that the hydraulic conditions of a given installation are simulated by CFD, while the trajectory of particles seeded in the flow are solved by a DEM solver. A coupling algorithm describes the interaction between fluid and solid particles. Each particle is described individually and acts in the application at hand as a fish surrogate. The DEM solver allows for a tracking of the contact interactions of particles with their environment, more specifically the inter-particle contacts and contacts between particles (i.e., fish) and wall boundaries. For the risk assessment model at hand interactions between particles are of no importance and the focus is laid on wall interactions, as they describe blade strike events in hydraulic installations and in pumps or turbines. Adding to the CFD-DEM model a complex fish behaviour model is the next logical step to improve prediction accuracy. The collision prediction capabilities of the CFD-DEM simulations are combined with rules of conduct for individual particles derived from data obtained by ethohydraulic experiments on live fish. It is obvious, that an important role in the development of such a prediction tool is the validation with experimental data from laboratory or field tests.

Data from field tests exist for the Las Rives hydropower plant. In this study it serves as an example of those anthropogenic barriers mentioned above. The plant is located on the Ariège river in France. To increase the performance of the site a dam was built to elevate the water intake. Therefore, a fish ladder for upstream migration and a bypass system for downstream migration were installed in the facility to maintain river continuity. The upstream fish passage is located at the upstream end of the dam and the downstream fish bypass is located after a rack in front of the intake channel of the power plant. The downstream fish bypass later connects to the tailwater by a free-stream jet. The objective of the original experiments was to investigate the injury risk for the fish in a downstream passage of the facility, caused by impacts during water surface penetration or by hitting the river bed. In France, the only currently available criterion for such a water jet bypass system has been proposed by Odeh and Orvis, 1998 [11]; it states that a minimum water depth of one quarter of the fall height – with a minimum value of about 1 m – has to be maintained. This expression omits other important parameters like the jet velocity at the impact point, which depends on the initial flow velocity at the outlet of the bypass, and the flow rate, which determines the thickness and the penetration depth of the jet.

The goal of this work is to develop a numerical model of the downstream fish passage. This model will form the basis for further investigations, where the numerical data will be compared to passive sensor probes and live-fish tests.

2. METHOD AND MATERIAL

2.1. Numerical model of the Las Rives installation

The dimensions chosen for the numerical model are based on the technical drawings of the Las Rives site. To keep acceptable computing times, only the important part of the installation is considered, i.e., the channel and the pool region. The channel is 4 m in length and 1 m wide, with a slope of 4°. The ground profile of the river is provided by measurements carried out on site. In the simulation setup, the pool/river basin has dimensions of 10 m × 10 m × 6 m [length, width, height]. The vertical drop from the channel outlet to the river ground is approx. 3.6 m and the entire numerical domain is shown in Figure 2.

The downstream fish bypass at the Las Rives hydropower plant site, which connects the intake channel to the tailwater by a free-stream jet is shown in Figure 1.

Figure 1. The water jet leaving the downstream fish passage into the underlying river at the Las-Rives-Site

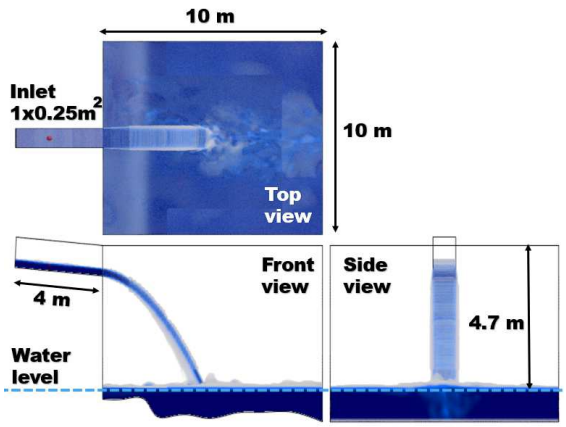

Figure 2. Technical sketch of the numerical domain.

2.2. Numerical set up

The numerical simulations were built with use of the Star-CCM+ software package, while the two-phase flow was described by an Eulerian multiphase model using the Volume of Fluid (VOF) method [12]. The VOF method is a simple multiphase flow model able to resolve the interface between the phases of the mixture by adding a volume fraction transport equation. For two phases only one volume transport fraction is calculated. It is assumed, that no interface interaction occurs and therefore all phases share the same velocity, pressure, and temperature fields. A segregated SIMPLE (Semi-implicit Method for Pressure Linked Equations) solver was chosen. The incompressible form of the Reynolds-Averaged Navier-Stokes (RANS) equations were deployed to capture the highly turbulent, fluctuating flow with reasonable computational costs. More sophisticated, but also much more expensive methods, like Large-Eddy Simulations (LES), were not considered, since many operating points of the device shall be investigated in future work. Based on preliminary studies, the two-equation $k-\omega$-SST (shear stress transport) approach introduced by Menter was chosen for turbulence modeling [13], where k is the turbulent kinetic energy and ω is the specific rate of dissipation. The quality of this model has been demonstrated in many applications, but it may still underpredict anisotropy effects, and as a consequence the flow field [14]. The boundary layer is treated in an adaptive wall model depending on the y^+ criteria. y^+ is the non-dimensional wall distance. Overall, the numerical model described in Table 1 appears as a good trade-off between numerical robustness, computational costs, and resulting accuracy. Simulations were run in parallel on 25 nodes (16 core- Haswell), corresponding to 400 CPUs on the high-performance Linux cluster Sofia of the University of Magdeburg, progressing at a rate of approx. 0.12 CPU hours per simulated second of physical time. One CFD simulation requires approximately 960 CPU-hours in total to reach a quasi steady state, which corresponds to 2.4 hours.

Table 1. Overview of numerical set up.

Solver physics	3d, implicit unsteady, incompressible, turbulent,Eulerian -multiphase, segr. flow
Solver algorithm	SIMPLE
Temporal discretization	
Accuracy	first order
Time step	$1 \cdot 10^{-3}$ s
Spatial discretization	
mesh type	core: trimmed cells, near-wall region: orthogonal prism cells
core cell size:	0.1 m
refinement cell size:	0.05 m
finest cell size:	0.04 m
Total amount of cells	$\approx$ 2.4 million
CFL (target)	< 0.5
Turbulence model	(Menter) $k-\omega$ SST
Wall treatment model	adaptive
Multiphase model	Volume of fluid (VOF)
Convergence criteria	
Residuals determined by	RMS
X-Momentum equations	residual target= 0.005
Y-Momentum equations	residual target= 0.005
Z-Momentum equations	residual target= 0.005

2.2.1. Boundary condition

The inlet of the numerical domain is implemented as velocity inlet condition. The velocity of the bypass is given by experimental data; velocity and flow direction are specified. Inlet volume flux as well as the fluxes of momentum and energy are subsequently calculated. The pressure condition at the inlet is set to zero gradient. To model the interactions with the environment (atmosphere and domain outlet), a pressure outlet boundary condition is set. This boundary

pressure can be considered as the static pressure of the environment (here, static pressure of air and water) into which the fluid enters. The boundary face velocity is extrapolated from the inner domain and the static pressure. The water height level of the underlying river is controlled by setting the pressure at the pressure outlet boundaries to the hydrostatic pressure, which is calculated by the predefined depth of water. The components of the volume fractions at the pressure outlet boundaries are also set in accordance to the depth of the water. This definition sets the boundaries equal to an indefinite extended basin with constant water level. However, this assumption isn't fully in accordance with the real river flow direction, which runs *perpendicular* to the inlet chamber. However, the effects regarding the interest of the study, a fish injury risk assessment, were considerd to be negligible. An overview of the boundary conditions can be seen in Figure 3.

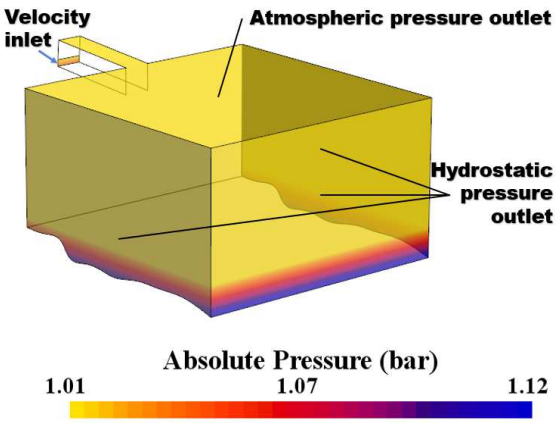

Figure 3. Sketch of the numerical domain and its boundary conditions. The inflow direction is displayed by the blue arrow at the channel inlet.

2.2.2. Meshing

The discretization of the core domain is achieved by a hexahedral mesh, whereas the near-wall region is generally meshed with a moderately refined boundary layer according to the upper limit range of the adaptive y^+-based near-wall treatment ($30 < y^+ < 100$). The centroid of the near-wall cell lies in the log-layer of the boundary layer. A full resolved boundary layer was considered to be to computational costly and not necessary for the study at hand. Using the VOF model comes with certain stability requirements for the spatial and temporal discretisation. One of those best-practice guidelines from the software provider relates to the Courant-Friedrich-Lewy number (CFL). The CFL number is a non dimensional number to evaluate the spatial and temporal resolution of a CFD simulation. It relates the time scale for the flow through a mesh cell to the time step size. A value smaller than one means that the flow 'remains' in one cell in between two timesteps, which is a convergence criteria for explizit formulated numerical schemes. An increase of the value over one will lead to inaccuracies and instabilities in this case. For simulations applying the VOF-model, the CFL number should be less than 0.5 following the code documentation of the software used in this study.

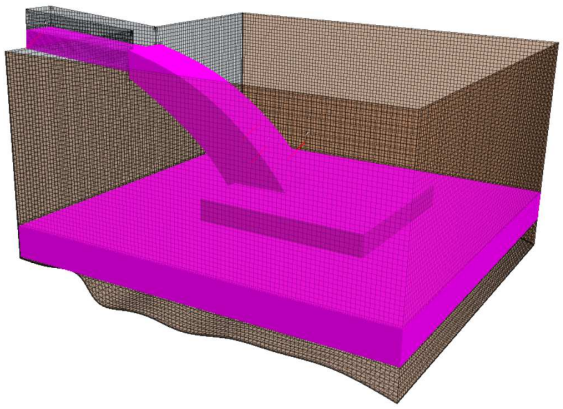

Figure 4. The area of mesh refinement in the simulation domain is highlighted in pink

To follow the computational requirements, a refinement area is defined (see figure 4). It contains the water jet and its impact on the water surface, down to the river bed. This can be seen even more clearly in figure 5. The time step is set to $1 \cdot 10^{-3}$ s, ensuring stability.

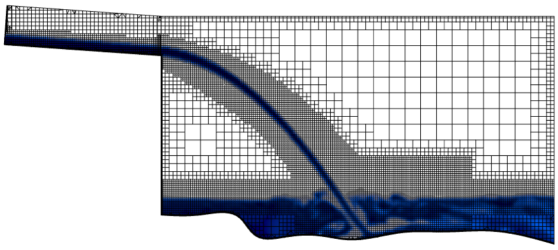

Figure 5. Cross-section through the middle of the domain, showing mesh size and two-level refinement areas. The first is located at the air/water boundary while the second, even finer refinement is placed at the location of the jet/river bed interaction

2.2.3. Grid independence study

To ensure grid independency, a mesh size study was performed. The refinement in this study only takes place in the area, which contains the water jet and the area around the place where the jet hits the bottom. This is chosen, for the aim of the study is to show the impact of objects hitting the water surface and the river bottom through the jet. The refinement has been carried out in three levels. The first

level is coarse with a cell size of 0.1 m, the second refinement level has a cell size of 0.05 m and the third/finest refinement level has a cell size of 0.04 m. The three refinement levels are compared on a velocity sample placed in the middle region of the jet stream. The placement of the velocity sample is shown in Figure 6 in a top and side view.

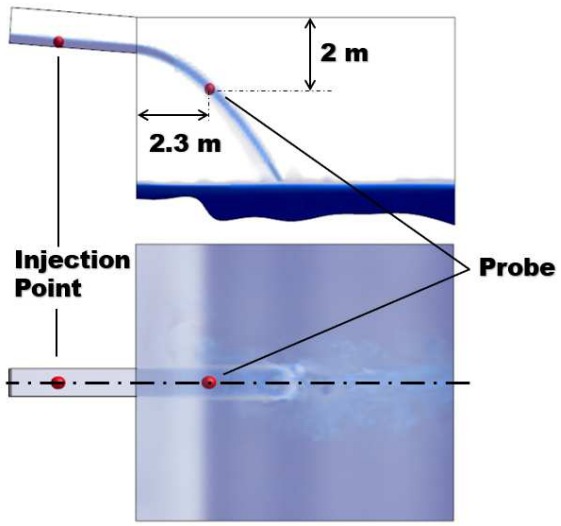

Figure 6. Probe and injection point location for grid independence study in water jet stream

The comparison of the refinement levels is shown in Figure 7 . Here, the averaged velocity at the sample point is plotted against the number of cells in the simulation area. The averaging of the velocity takes place over a period of 5 s of physical time and after the simulation has reached a quasi static state (after around 15 s of physical time). Considering the computational cost and the grid independency the cell size of the fine (1.6M cells) refinement level is decided to be sufficient enough for the present work.

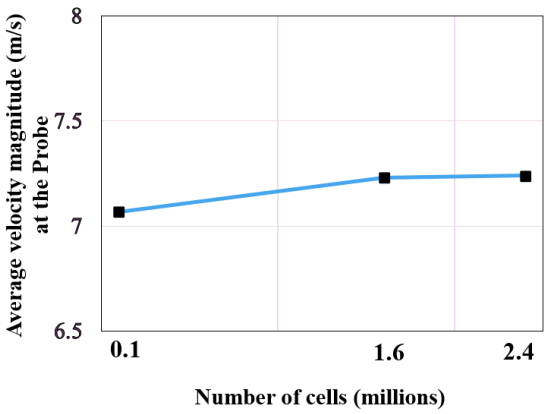

Figure 7. Grid independence study by average velocity magnitude at sample point over number of cells for three different refinement levels.

2.2.4. Domain size independence study

In addition to the grid independence study, a domain size independence study has also been conducted. The topology of the river bottom is only known for a quite limited area. The piercing of the the water surface by the jet and the subsequent hitting of the river bed results in a complex two-phase flow with entrained air. This flow is dominated by the air entrainment effects and the redirection of the jet at the river bed. The main direction of flow is in the x-direction which convects these disturbances in this direction. Due to the short distance to the outlet, the flow remains chaotic with back flow regions at the boundary of the domain in the initial configuration. The simulation domain boundary was iteratively extended (in x-direction) to investigate the influence of the domain size.

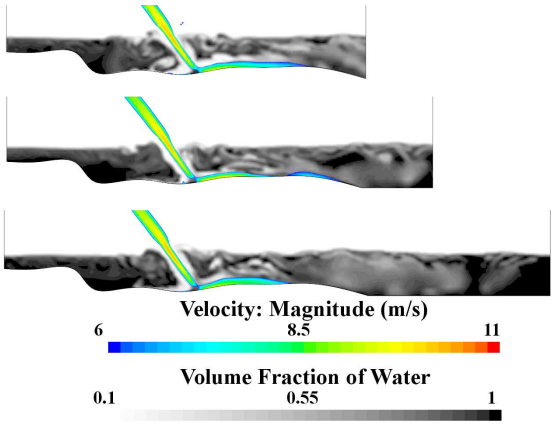

Figure 8. Domain size independence study of three different configurations showing the cross-sections in x-direction with velocity field of the liquid phase, from jet injection to its impact on the pool ground. Additionally the gray-scaled area shows the local volume fraction of the water

Figure 8 shows the influence of the extension of the simulation domain. The velocity field of the jet (colored) and volume fraction of water (black and white) are depicted. The first two variants still feature a strong mixing of air and water up to the boundary limits. The widest simulation domain shows a homogeneous flow field and the outlet flow consists entirely of water. This allows for a stable simulation. In consequence the last setting is chosen for further investigations.

2.3. Active particles as fish surrogate in CFD-DEM

The overall goal in this study is the investigation of injury and mortality risk of fish in the bypass by an extension of the established CFD model with a coupled DEM method using active and passive particles.

The particles are described in a Lagrangian framework, which means that each path of an in-

jected particle is tracked when travelling through the domain. The continuous phase is described in an Eulerian framework and the combination of both leads to an Eulerian-Lagrangian multiphase flow. The deployed Eulerian-Lagrangian multiphase model calculates the trajectory of each particle and the momentum equations for the fluid flow – taking all interactions between the dispersed phase and the continuous phase into account. Different coupling levels are possible, in which only flow to particle information transfer is considered (one-way-coupling), or additionally the feedback of the particle onto the flow (two-way-coupling). Choosing the DEM model as a Lagrangian representation of the particle adds two more coupling levels, when taking into account particle-particle and/or particle-wall interaction. For this project the particle-wall interactions is of highest interest because it corresponds to fish (i.e., particle) – blade (i.e., wall) interactions. The DEM model enables the recording of contact forces for impacts, which can be used for prediction models regarding fish mortality and injury risk. Besides using generic, passive particles as fish surrogates a novel approach with active particles, first introduced in [15] will be deployed as well. The intention of the novel approach is an active motion of the particles according to a fish-like behavior, based on ethohydraulic observations. The method adds an additional body force to the particle equation of motion, with which the velocity and orientation of the particle can be controlled. Here, different rules of conduct for the particle can be defined and combined to mimic real fish's behavior. Such rules of conduct can be for instance: to follow the instantaneous main flow, the willingness to move downstream/upstream, or to swim close to the wall or to the ground. A detailed description of the method can be found in Powalla et al. 2021 [15].

In figure 9 an example (see [16] for details) of the active particle approach is shown. In this example, two rules of conduct are acting on the particles. One is the willingness to migrate downstream the power plant. The other is to follow the main flow. They are both linked by a weighting function, which controls the impact of each rule of conduct. Their combination results in a final orientation of the particle and can be combined with individual weights for each particle.

3. RESULTS

In the application at hand, specific challenges in the numerical model are found in the region where the water jet enters the river and subsequently, when the jet hits the river bed. Here, the propagation of strong waves caused by the impact of the jet on the surface has to be captured. Even more complex is to capture the air entrainment effects. Figure 10 shows the water phase for the entire domain. It can be seen how the water mixes up with the air when piercing the water surface. It is also visible that surface waves travel toward the domain boundaries, which causes further challenges for stability and accuracy.

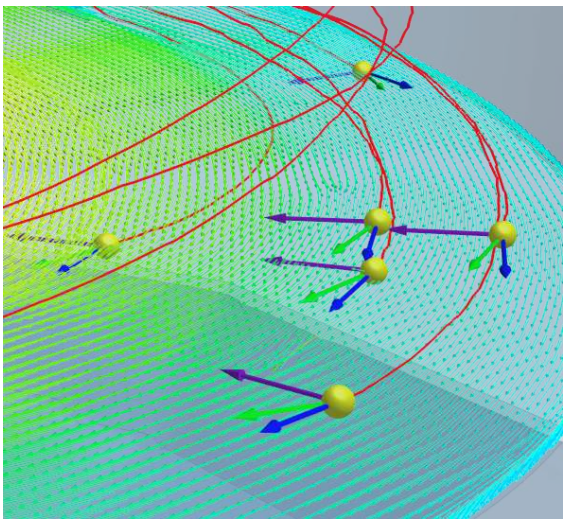

Figure 9. Example showing the active particle method. Two rules of conduct guide the particles (yellow): follow the stream (blue vector), and willingness to migrate downstream (purple vector). The combination of both results in the final orientation (green vector).

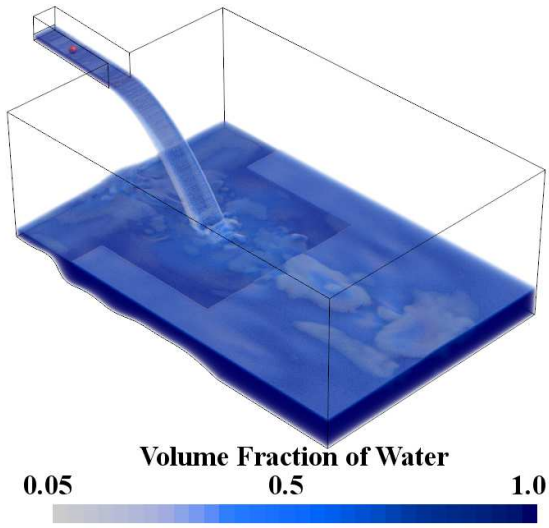

Figure 10. 3D Simulation of the water jet displaying the liquid phase (defined by volume fraction of water > 0.5)

As discussed in the meshing section, the VOF method requires a certain temporal and spatial resolution. An indication for that can be seen by the CFL number. Figure 12 shows the CFL number for the water phase. It can be seen, that it ranges for the main jet stream in between 0 to 2. This is above the best-practice limit of 0.5, due to locally large velocities (see for comparison figure 11). However, the spatial resolution of the simulation is found to be sufficient in most areas.

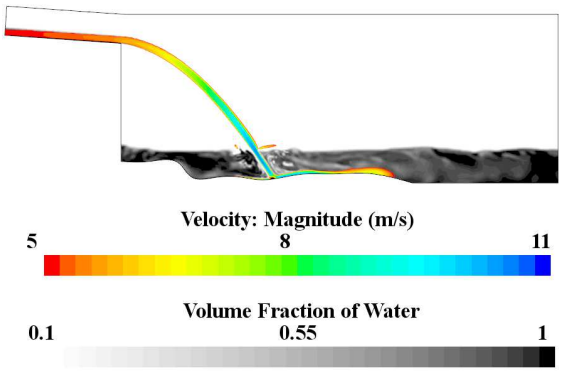

Figure 11. Cross-section with velocity field of the liquid phase, from jet injection to its impact on the pool ground. Additionally the gray-scaled area shows the local volume fraction of the water.

The transport of the air and water mixture by the jet into the pool is highlighted in figure 11. The figure clearly shows the highly turbulent and complex fluid structure not only caused by the multiphase mixture but also by the impact of the jet onto the river bed. It also shows the acceleration of the flow from leaving the top channel to the bottom of the pool. Here, the flow accelerates from its initial value of 5 m/s up to about 11 m/s.

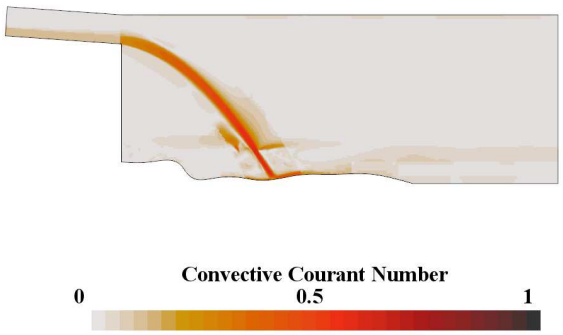

Figure 12. CFL number of the liquid phase in the central cross-section, shown together with mesh refinement.

In figure 13 a first simulation involving particles is shown. The particles are injected at the level of the inlet channel and are transported by the jet into the underlying water basin where they finally hit the ground. The aforementioned method allows to extract the acceleration and contact data of each particle which will be used ultimately to quantify risk injury for fish in such a bypass system.

4. CONCLUSION AND OUTLOOK

In this ongoing work the numerical model of a water jet from a bypass facility for downstream fish

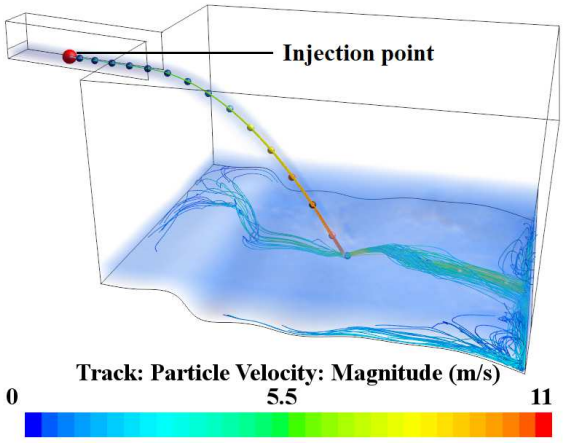

Figure 13. First simulation with particle injection. The injection is at the channel inlet (red spheres) and the particles are then colored according to their instantaneous velocity.

passage has been presented. This model describes both hydrodynamic features and fish behavior. The piercing of the water surface by the jet and the subsequent hitting onto the river bed results in a complex two-phase flow, which leads to a numerical challenging task. In this work a grid independence study and a domain study has been conducted to ensure numerical stable conditions for the upcoming tasks. The assessment of the injury risk for fish passing the bypass structure. It has to be mentioned that in the current setting the flow velocity of the underlying river is not yet considered and the influence has to be discussed and addressed in the future

In the next steps, this model will be used to quantify injury and mortality risk of fish in the downstream passage of the bypass, using the formerly introduced procedure combining CFD/DEM with active fish behavior [15]. The particles will be tracked during their path through the domain and the data will be validated with use of experimental measurements gained by passive probes [17] and live fish tests.

ACKNOWLEDGEMENTS

The authors are grateful for the support of Sylvie Tomanova and Dominique Courret from the Office français de la biodiversité (OFB) in sharing data and insights of the Las Rives site necessary for the realization of the study at hand.

REFERENCES

[1] Silva, A. T., Lucas, M. C., Castro-Santos, T., Katopodis, C., Baumgartner, L. J., Thiem, J. D., Aarestrup, K., Pompeu, P. S., O'Brien, G. C., Braun, D. C., Burnett, N. J., Zhu, D. Z., Fjeldstad, H.-P., Forseth, T., Rajaratnam, N., Williams, J. G., and Cooke, S. J., 2018, "The Future of Fish Passage Science, Engineering, and

Practice", *Fish and Fisheries*, Vol. 19 (2), pp. 340–362.

[2] EP, 2000, "Directive 2000/60/EC of the European Parliament and of the Council of 23 October 2000 Establishing a Framework for Community Action in the Field of Water Policy", .

[3] Trumbo, B. A., Ahmann, M. L., Renholds, J. F., Brown, R. S., Colotelo, A. H., and Deng, Z. D., 2014, "Improving Hydroturbine Pressures to Enhance Salmon Passage Survival and Recovery", *Reviews in Fish Biology and Fisheries*, Vol. 24 (3), pp. 955–965.

[4] Bevelhimer, M. S., Pracheil, B. M., Fortner, A. M., and Deck, K. L., 2017, "An Overview of Experimental Efforts to Understand the Mechanisms of Fish Injury and Mortality Caused by Hydropower Turbine Blade Strike", *Technical Report, US Dpt of Energy*.

[5] Giesecke, J., Heimerl, S., and Mosonyi, E., 2014, *Wasserkraftanlagen: Planung, Bau und Betrieb*, Springer Vieweg, Berlin, 6., aktualisierte und erw. aufl edn., ISBN 978-3-642-53870-4.

[6] von Raben, K., 1957, "Zur Frage der Beschädigung von Fischen durch Turbinen", *Die Wasserwirtschaft*, Vol. 4, pp. 97–100.

[7] Montén, E., 1985, *Fish and turbines: fish injuries during passage through power station turbines*, Vattenfall.

[8] Turnpenny, A., Clough, S., Hanson, K., Ramsay, R., and McEwan, D., 2000, "Risk assessment for fish passage through small, low-head turbines", *Final Report, Energy Technical Support Unit, Harwell, UK*.

[9] Larinier, M., and Travade, F., 2002, "Downstream migration: problems and facilities", *Bulletin Français de la Pêche et de la Pisciculture*, pp. 181–207.

[10] Ebel, G., 2008, *Turbinenbedingte Schädigung des Aals (Anguilla anguilla) : Schädigungsraten an europäischen Wasserkraftanlagenstandorten und Möglichkeiten der Prognose*, Ebel, Büro für Gewässerökologie und Fischereibiologie, ISBN 9783000254451.

[11] Odeh, M., and Orvis, C., 1998, *Downstream fish passage design considerations and developments at hydroelectric projects in the northeast USA*, Fishing News Books.

[12] Hirt, C. W., and Nichols, B. D., 1981, "Volume of fluid (VOF) method for the dynamics of free boundaries", *Journal of Computational Physics*, Vol. 39 (1), pp. 201–225.

[13] Menter, F. R., 1994, "Two-equation eddy-viscosity turbulence models for engineering applications", *AIAA journal*, Vol. 32 (8), pp. 1598–1605.

[14] Wallin, S., and Johannsen, A. V., 2000, "An explicit algebraic Reynolds stress model for incompressible and compressible turbulent flows", *Journal of Fluid Mechanics*, Vol. 403, p. 89–132.

[15] Powalla, D., Hoerner, S., Cleynen, O., and Thévenin, D., 2022, "A Numerical Approach for Active Fish Behaviour Modelling with a View Toward Hydropower Plant Assessment", *Renewable Energy, (in press)*.

[16] Powalla, D., Hoerner, S., Cleynen, O., Müller, N., Stamm, J., and Thévenin, D., 2021, "A Computational Fluid Dynamics Model for a Water Vortex Power Plant as Platform for Etho- and Ecohydraulic Research", *Energies*, Vol. 14 (3), p. 639.

[17] Tuhtan, J., Fuentes, J., Angerer, T., and Schletterer, M., 2018, "Monitoring upstream passage through a bypass pipe and drop at the fish lift Runserau: Comparing dynamic pressure measurements on live fish with passive electronic fish surrogates", *12th International Symposium on Ecohydraulics Aug19 Aug24,2018, Tokyo, JAPAN*.