Particle-based evaluations of fish-friendliness in Kaplan turbine operations

P Romero-Gomez, M Lang, J Michelcic and S Weissenberger
Research and Development, ANDRITZ HYDRO GmbH. 4031 Linz, Austria
E-mail: pedro.romero-gomez@andritz.com

Abstract. The concept of fish friendliness of hydroturbines relates to the effects of a unit design and its operating conditions on the success of migratory fish survival during passage through turbine flows. Most computer-based studies of fish friendliness approximate fish trajectories with streamlines; this work introduces a particle-based method to account for collisions on two typically interrogated structures: the stator and the runner. We implemented the method in various operating points of a Kaplan turbine simulated with computational fluid dynamics. The results indicated that only a small percentile of the particle sample collides severely with the stator (<0.5%) and the runner blades (<3%). The severe collisions did not exhibit any preferential distribution in the circumferential and radial directions, except for particles to collide near the outer area of the runner (near the discharge ring). This article concentrates mainly on the advantages and limitations of the particle-based method, always assessed in view of streamline-based calculations.

1. Background
The stakeholders involved in constructing and operating hydropower stations have long recognized the environmental consequences of hydropower generation, particularly the detrimental effects on migratory fish survival. Based on this recognition, researchers and practitioners have developed expert knowledge for better understanding the hydraulic conditions leading to fish mortality and for designing systems to ameliorate the ecosystem disruption affecting fish populations. Among the measures to increase fish survival onsite, the retrofitting of runner designs to make them more “fish friendly” has attracted considerable attention (extensive summaries in [1–3]). Field studies of fish friendliness, for instance, have deployed sensor units ([4]) that record hydraulic conditions during passage through turbine flows, thereby informing us about extreme conditions that fish may encounter (one extensive field test was reported in [5]). Such field tests are essential to gain understanding of the phenomena affecting fish survival but can be carried out only after the turbine is installed and in full operation. Computer-based methods, on the other hand, facilitate the integration of desirable geometric features during the unit design phase. This study introduces a modeling method for evaluating fish friendliness using Lagrangian particles and implements it in a recent turbine rehabilitation project.

The work in reference [6] laid the conceptual ground for a variety of recent computer-based studies for fish-friendliness of hydroturbines. Such studies consist mainly of a flow description with Eularian approaches that support the simulation of potential fish trajectories based on Lagrangian agents moving within the 3D flow fields. For instance, evaluations of fish friendliness in turbine units of the Columbia River (U.S. Pacific Northwest, see reference [7, 8]) were based
on flows simulated with computational fluid dynamics and fish trajectories calculated with streamlines, which intrinsically assumed that fish were passively carried through the turbine flows while neither body nor surface forces acted on the fish body. However, streamlines do not collide with walls and preclude in this way the resolution of any collision event. In this work, particles are proposed as an alternative method to overcome the aforementioned limitation of streamlines.

Previous studies have already contributed to describe the forces and fluid-body interactions that need to be accounted for to represent fish bodies as inertial particles moving in highly turbulent flows arising from hydraulic machinery [9–11]. The present study builds up on such knowledge to evaluate collision rates on guide vanes and rotating blades, as well as low pressures at various operating points using Lagrangian particles. We focused our study on the exposure of fish to hydraulic stressors; we purposely defer the analysis of the biological consequences of such events for future work.

2. The Eddersheim Dam
2.1. Site and Turbine Unit Description
The Eddersheim lock-and-dam structure is located in the “Main” River at kilometer 15.55 and operates nominally with a gross head of 3.61 m and total design discharge of 180 m$^3$/s. Its forebay water surface elevation is at 87.53 masl. It consists of three Kaplan-type turbines of nominal 4.1 meters of diameter, four blades each, operating at a rotational speed of 75 rpm with a unit rating capacity of 1.8 MW. The dam project was completed in 1941 and put in operation in 1942.

The Eddersheim dam belongs to a system of hydropower stations that have historically contributed to a decline in fish populations, and have therefore implemented a series of measures to increase passability and survival of downstream migratory fish. In 2017, ANDRITZ Hydro was commissioned to design and replace the original Kaplan turbine units. One of the essential features of the new turbines must be a reduction in the mortality rates of migratory fish species, most importantly salmonids and eels. For this purpose, the computer-based method described in reference [12] was successfully implemented to evaluate the fish friendliness of various operating points in the original turbine and to test the enhancement of the new turbine over the original one. Such method is cast in a software suite called the Biological Performance Assessment tool (the BioPA, developed at the Pacific Northwest National Laboratory, Richland, WA, U.S.A.) and defines a global score for each of the four hydraulic stressors accounted for in the comparative analysis of hydroturbines: strike probability, pressure, turbulence and shear flows. In the present application, we focused on the first two (collision rates and nadir pressures) and conducted the evaluations with the use of Lagrangian particles—instead of using streamlines to simulate the fish trajectories as the BioPA does.

2.2. Selection of analyzed operating points
The Kaplan turbine is a double regulated machine which allows us to operate for one head and discharge with a different configuration of the runner blade opening ($\beta$) and guide vane opening ($\alpha$). The idea was to operate the machine not only at the best efficiency point, because an operation at off-cam conditions can influence the fish collisions. For this reason, three runner blade openings where chosen with the guide opening which lead to same discharge conditions at maximum head ($H_{\text{max}}$). These three operating points are called A, B, C in this paper (figure 1). Another opportunity to influence fish impact in a powerhouse is by changing the number of turbines in operation. The plant layout is usually designed to manage the river discharge with all turbines at full load conditions. In times of low water, fewer turbines can manage the river discharge at full load or more turbines can operate at part load. In our case, we tested one turbine in operation at full load which has to manage the complete river water and two turbines...
in part load, which have only to operate for half of the river water. The operating point for these two turbines with smaller runner and guide vane openings is called D in figure 1.

3. Modeling approach

3.1. Goals and Conceptual Description

A number of mechanical and hydraulic stressors causing fish mortality have been described in specialized summaries in this field ([1, 3]). In this work, we analyzed (i) the occurrence and intensity of collisions at various operating points and (ii) the lowest (nadir) pressures that may cause barotrauma—a pressure-related injury that fish usually experience due to rapid decompression during turbine passage. To do this, we combined various modeling techniques to understand the physical processes influencing the potential trajectory of migratory fish through the turbine units. Such techniques are shown as distinctive elements of figure 2. The geometric model of the hydroturbine unit was discretized into finite volumes (element 1) over which the conservation equations were solved. The numerical solution to such equations constitutes the flow conditions (element 2) through which fish (particles) move after they are released from prescribed locations. The particle trajectory (element 3), in turn, governs the occurrence of collisions on interrogated boundaries that are typically subject to design/modifications in turbine replacement projects, e.g. stay vanes, guide vanes and runner blades (element 4). Equally important to detecting such collisions is the recording of the hydraulic conditions along the trajectory. The inset labeled as element 5 in figure 2 exhibits the time series of acceleration magnitude (in g) and absolute pressure (in kPa) for the trajectory labeled a element 3. The signal peaks mimic the corresponding measurements with autonomous sensor devices, i.e., collisions in the field are not directly recorded but instead estimated with the help of post-processing algorithms that implement criteria on the duration and magnitude of pressure and acceleration recordings. Large particle sets were released so that the results are statistically meaningful (element 6).

3.2. Flow simulations with computational fluid dynamics

For the most part, we followed the long consolidated industry practices for modeling of hydropower turbine flows with computational fluid dynamics (CFD). However, two aspects were unique to this application: the solution at prototype scale (design is always done at physical model scale) and the inclusion of the full stator and runner geometries (360°) instead of a single flow passage sector. The volumetric computational domain was discretized with 28 M polyhedral cells, a meshing technology available from the software used [13]. The unit geometry consisted of the intake (7.8 M cells), stator (8.5 M), runner (4.7 M) and draft tube (7.0 M). The meshing strategy incorporated refinements near solid boundaries, with considerable refinements near the guide vanes and runner blades that resulted in wall $y^+$ values of 25.7 and 13.3, respectively.

Flows were solved in steady state mode using the realizable $\kappa-\epsilon$ turbulence model with an “all $y^+$” wall treatment selected from the available software solvers. The flow solver was segregated (i.e., one equation was solved for all cells first, then the next equation followed suit) and the convection scheme was 2nd-order. The boundary condition was set as a differential pressure between the inlet and outlet equal to the gross head (3.41 m), which is very constant in the site operations. The runner rotation was modeled with a local reference frame with an angular
speed of 75 rpm. The interfaces between stationary and rotationary regions were selected as “mixing planes”, which implemented a circumferential averaging of the incoming flow field data towards the outgoing interfacing boundary. This interfacing selection is equivalent to the “stage interface” in other commercial CFD softwares (CFX).

Similar to all CFD flow simulations of this kind, the solutions consisted of the three dimensional flow fields of velocity (three components), pressure, turbulent kinetic energy ($\kappa$) and its dissipation rate ($\omega$). These conditions constitute the primary source of forcing on the Lagrangian particles that represented fish bodies moving within the flow and colliding with the interrogated structures.

3.3. Particle simulations with a Lagrangian approach

The Lagrangian particle tracking technique consists of solving for the velocity of spherical particles ($\bar{\vec{u}}_p$ in equation 1) as they are influenced by the 3D flow conditions solved with CFD. Lagrangian spheres have long been used in engineering applications that closely resemble the present study, e.g., erosion wear of hydroturbine runners [14], particle deposition in gas turbines [15], and bird collisions on aircraft structures [16]. Two challenges are notorious in the implementation of Lagrangian particles. First, the definition and formulation of forces acting on the particles largely depend on both the particle size and the turbulent scales in the flow field. In this application, we are concerned with the so-called inertial particles, which are characterized by a considerably large response time with respect to the viscous damping rate (i.e., large Stokes number). Second, particles passing through hydroturbines cross a variety of flow conditions that may give rise to distinctive fundamental responses of one particle along its trajectory. Previous studies sought to address the aforementioned aspects and implemented

Figure 2. Conceptual description of the modeling approaches used in this work: (1) meshing, (2) flow simulation, (3) Lagrangian particle tracking, (4) collision detection, (5) data recording and (6) large particle samples.
particle tracking formulations in conventional hydro- and hydrokinetic turbines [9–11].

The particle equation of motion dictates that the change in momentum (left-hand side of equation 1) of a particle of mass $m_p$ is influenced by surface and body forces. The drag force ($\bar{F}_d$, defined in equation 2) is a function of the instantaneous slip velocity (fluid velocity minus particle velocity, or $\bar{u}_s = \bar{u} - \bar{u}_p$), the surface area of the sphere ($A_p$) and the drag coefficient ($C_d$), that followed the Schiller-Naumann formulation. This term accounts for the small-scale features that cannot in practice be resolved, but the effects of which have been measured in empirical studies. When large localized pressure gradients are present, $\bar{F}_p$ (equation 3) has a strong effect on the particle volume ($V_p$), which is the case in the interblade regions where static pressures ($p_{static}$) change considerably from the pressure- to the suction-sides of the blades. The term $\bar{F}_{vm}$ (equation 4) relates to the volume of the surrounding fluid that gets displaced or deflected owing to the particle motion. $C_{vm}$ is equal to 0.5 for spheres moving in incompressible flows. The shear lift force ($\bar{F}_{ls}$) acts whenever there exists a velocity gradient in the direction orthogonal to the relative motion of the particle, and follows the formulation in reference [17]. Finally, the dispersive effects of turbulence on the particle were accounted for by applying a random walk technique based on the turbulent fields [18]. The fundamental premise of such method stipulates that apart from the mean flow field, particles experience a velocity fluctuation ($\bar{u}'$) contained in the eddies they cross. Further details of the turbulent dispersion model and the other forces affecting the particle motion were provided in the reference [11].

$$m_p \frac{d\bar{u}_p}{dt} = \bar{F}_d + \bar{F}_p + \bar{F}_{vm} + \bar{F}_{ls}$$  \hspace{1cm} (1)

$$\bar{F}_d = \frac{1}{2} C_d \rho A_p |\bar{u}_s|$$  \hspace{1cm} (2)

$$\bar{F}_p = -V_p \nabla p_{static}$$  \hspace{1cm} (3)

$$\bar{F}_{vm} = C_{vm} \rho V_p \left( \frac{D\bar{u}}{Dt} - \frac{d\bar{u}_p}{dt} \right)$$  \hspace{1cm} (4)

3.4. Severe collision detection and nadir pressure calculation

Modeling Lagrangian particles require a definition of their behavior as they impinge on a solid boundary. The selection herein allowed for particles to rebound on all walls they encounter during their trajectory. During a rebound event, the particle velocity was modified according to restitution coefficients that specify the proportion of the incoming velocity that gets transferred to the outgoing velocity. We assumed perfectly elastic rebounds, the restitution coefficients of which are equal to 1.0 for both the velocity components tangential and normal to the wall. The rebounds and their conditions—location, impact velocity, angle—were recorded by means of a boundary sampling capability implemented in two prescribed boundaries of the domain: the guide vanes and the blades. The recorded collision data in combination with the trajectory data aided in the grading of the collision intensity based on the angle formed between the incoming and outgoing velocity vectors. This criterion is to a large extent based on the possibility that experimental capabilities may make it difficult to distinguish between mild collisions and deflected trajectories with obtuse angles ($\theta > 120^\circ$).

The nadir pressure ($P_{nadir}$, equation 5) corresponds to the minimum value of the absolute pressure ($P_{abs}$) that each particle encountered along its trajectory. The recorded history of pressure ($P_{part}$) was adjusted by accounting for the hydrostatic pressure [$g \rho (z_0 - z_{part})$, where $z_0$ is the tailrace water surface elevation and $z_{part}$ is the particle centroid elevation] as well as the atmospheric pressure, $P_{atm} = 101,300$ Pa. $P_{correction}$ refers to the surface-averaged value of $P_{part}$ at the model outlet which lies relatively distant from the runner in the simulated domain.
\[ P_{\text{nadir}} = \min \{ P_{\text{abs}}(t_i) \} = \min \{ P_{\text{part}}(t_i) + g \rho (z_0 - z_{\text{part}}(t_i)) - P_{\text{correction}} + P_{\text{atm}} \} \]  

4. Implementation and results

4.1. Implementation

As explained in section 3.2, flows were solved in steady state in order to obtain mean conditions throughout the unit. Particles colliding on blades in rotating motion, however, need to be simulated in transient mode. To reconcile both modeling needs, the flow simulations were first completed until convergence was ensured and the flow solution was then “frozen”. After that, the overall solution was changed to implicit unsteady (time step, \( \Delta t = 4 \text{ ms} \)) so that the runner region could be set in rotating motion—thereby replacing the implementation of a local reference frame—and the particles were thereafter tracked. With this strategy, we ensured that the particle tracking proceeded in an efficient manner, although we recognize that the use of advanced turbulence modeling (e.g. detached or large eddy simulations) can significantly enrich the flow resolution, which in turn may influence particle trajectories. For the purpose of the present comparative study, the aforementioned approach sufficed satisfactorily.

Collision rates and pressures registered by particles (or autonomous sensors) strongly depend on the location from where they are released. Should the modeling results be compared against field or laboratory data, the particle release would have to follow the empirical injection strategy closely. In this comparative study, however, it sufficed to select an array of releases at the distributor entrance, with uniform intervals in the circumferential and vertical directions (figure 3) that gave a total (\( N_{\text{inj}} \)) of 788 locations. Over an injection period (\( T_{\text{inj}} \)) of 20 seconds, a subset of \( N_{\text{inj}} \) was randomly selected with a probability (\( P_{\text{inj}} \)) of 0.0004, thereby resulting on particle samples of 1,576 (\( \frac{N_{\text{inj}} \cdot P_{\text{inj}} \cdot T_{\text{inj}}}{\Delta t} \)).

The particle diameter (\( D_{\text{part}} = 4.36 \text{ cm} \)) corresponded to that of a sphere with volume equal to the cylindrical volume occupied by the autonomous sensor model with which field tests are usually conducted (\( V_{\text{cyl}} = L_{\text{cyl}} \cdot A_{\text{cyl}} = 9.6 \text{ cm} \cdot 4.524 \text{ cm}^2 = 43.4 \text{ cm}^3 \)). Particles were neutrally buoyant. In this way, we assumed that the trajectory of a spherical particle followed the inertial reaction of a cylindrical sensor; we recognized, however, that the motions of both geometric shapes—sphere and cylinder—are not the same. Work is in progress to eliminate this limitation. The selected time step allowed for two important time scales relevant in the collision detection scheme. First, the angular displacement of the runner blades were considerably small (1.8 °), which ensured no disruption in the flow conditions that particles experienced locally while transitioning from a stationary to a rotating region, and vice versa. Second, the particle track resolution at the moment of passage was high enough to not miss potential collisions in significant numbers. More details of the implementation can be found in references [19].

4.2. Results: collisions in the distributor

Collisions were identified and categorized for all operating point (see section 2.2). The distribution of the collisions is shown in figure 4 for the operating point C, and the remaining cases displayed similar patterns as the ones described in this section. For purposes of explaining
such patterns, four circumferential (I-IV) and three radial (leading edge, middle and trailing edge, or LE, MD and TE, respectively) zones were defined. Collisions exhibited a tendency to group in locations further into the spiral casing (zones III and IV) than in zones I and II. The reasons for that are related to the flow uniformity (or the deviation thereof) in the spiral casing. Releases in regions III and IV tended to experience longer travel distances through the stator, a circumstance that gave particles more physical space to depart from the ideal streamlines that theoretically would prevent collisions. Furthermore, passage channels in regions III and IV are typically associated with non-ideal flow conditions such as expansion of stagnation regions, extended recirculations and large pressure drops. Such undesirable phenomena are conventionally represented as a head loss value through the stator (figure 4b) and are known to promote particle entrainment, passage delays and collisions on near walls, i.e., disturbed flows cause wall contacts with higher frequency.

![Figure 4](image-url)  
**Figure 4.** Collision locations and zones (left) and contours of localized head loss (%) in the operating point C ($\alpha = 60^\circ, \beta = 27^\circ$)

Table 1 summarizes the most relevant information about the distribution and intensity of collisions for the four analyzed operating points. The collision rates tended to be low, with severe collision rates always below 1%. In the operating point C shown in figure 4, the severe collisions are shown in red. The severe collisions did not reproduce the aforementioned circumferential distribution; however, the very low severe collision rates presented herein do not allow for a statistically conclusive assessment and larger particle sample sizes are necessary. Irrespective of the operating point, most collisions occurred in the leading edge (zone LE) of the guide vanes.

| OP Label | $\alpha,^\circ$ | $\beta,^\circ$ | Collisions | Circumferential | Radial |
|---------|---------------|---------------|------------|----------------|--------|
|         |               |               | All, %     | Severe, %      | I      | II     | III    | IV     | LE    | MD    | TE     |
| A       | 50            | 29            | 4.5        | 0.38           | 23.9   | 22.5   | 36.6   | 16.9   | 63.4  | 28.2  | 8.5    |
| B       | 54            | 28            | 6.3        | 0.25           | 23.8   | 11.9   | 38.6   | 25.7   | 44.6  | 31.7  | 23.8   |
| C       | 60            | 27            | 6.1        | 0.50           | 10.3   | 36.8   | 34.0   | 19.6   | 55.7  | 26.8  | 17.5   |
| D       | 31            | 15            | 5.6        | 0.19           | 10.0   | 22.2   | 32.2   | 35.3   | 50.0  | 33.3  | 16.7   |

There were two patterns that turbine designers have long held for valid but did not manifest themselves in the present application. First, the collision rates did not increase with smaller values of $\alpha$, in other words, when gates are closing. It is widely believed that smaller guide vane angles would in turn limit the passage area for the particles, thereby increasing the potential for collisions. Second, our modeling results suggest that passively carried objects do not collide
with the trailing edge with high frequency, and that the instances when trajectories lie near the overhang of guide vanes are actually very rare. This observation is pertinent because there exists a persistent concern in the research community that fish are particularly vulnerable to collide and experience severe bodily damage when a gate overhang is present. Premises like these can be revisited with the use of particles to represent potential fish passages through the stator.

4.3. Results: collisions on the runner blades

Particles moving in the runner region (figure 5a) can either cross the inter-blade region without contacting the moving walls or register a collision, which leaves a sudden spike of certain magnitude and duration in the acceleration recordings. In this way, the present model-based calculations of particle collisions can be related to sensor fish data that are usually post-processed to determine the presence and intensity of collisions in prototypes [4]. In addition, collisions deflect the trajectory to various degrees, also allowing for grading the intensity of the contacts in a manner similar to the protocols in observational tests with physical models.

Figure 5. Trajectory (a), locations (b) and circumferential distributions (c) of collisions for operating point C ($\alpha = 60^\circ, \beta = 27^\circ$)

For operating point C (figure 5b), detected collisions exhibit a preferential location on the suction side of the blades. In the under-blade region, pressure gradients become large because a pressure recovery process takes place in relatively short timespan and space, which in turn influences the particle motion considerably (see equation 3). Such large pressure gradients promote wall contacts. Approximately 50% of the collisions lie on the leading edge of the blades (red in figure 5b), and that is the case for all operating points. There exists a tendency for collision rates to increase (see fourth column in table 2) with increasing blade angle ($\beta$), but such effect is not visible for those contacts categorized as severe, the collision rates of which remain under 3% in all cases with the same discharge. Laboratory and field tests of fish friendliness closely examine leading edge collisions because these are usually assumed to be the leading cause of mortal injury during passage. For that reason, turbine designers have also paid special attention on such geometric construction in the process of integrating potential fish friendlines enhancements. The low discharge operating point (point D) promoted all and severe collisions substantially. Low passage velocities derived from low discharges are known to also increase collisions detected with sensor fish data in prototype units [5, 20]. In such cases, not only the velocity plays a role, but also the non-optimal flow conditions arising from operations at part load.

For this particular turbine unit, the circumferential distribution of the leading edge collisions (figure 5c) neither relates to that of the stator for the same operating point (figure 4a) nor shows a particular behaviour across operating points. Three radial zones were also defined, each containing 1/3 of the runner passage area. All operating points show a clear tendency to push collisions towards the discharge ring owing to the effects of centrifugal forces. Observations of
this kind are relevant for turbine designers to increase the certainty in selecting design measures to improve fish friendliness in rehabilitation projects.

| Table 2. Summary of collision results in the runner |
|-----------------------------------------------|
| Operating point | Collisions | Only severe on each blade | Only severe radial dist. |
| Label | α | β | All Severe | Severe on LE | I | II | III | IV | % | % | % | % |
| A | 50 | 29 | 9.8 | 2.57 | 1.19 | 36.9 | 21.0 | 21.0 | 21.1 | 36.8 | 21.1 | 42.1 |
| B | 54 | 28 | 8.0 | 2.69 | 1.87 | 36.7 | 20.0 | 16.7 | 26.6 | 26.7 | 10.0 | 63.3 |
| C | 60 | 27 | 6.9 | 2.26 | 1.63 | 23.1 | 26.9 | 11.5 | 38.5 | 34.6 | 15.4 | 50.0 |
| D | 31 | 15 | 30.3 | 15.2 | 3.07 | 30.6 | 30.6 | 26.5 | 12.2 | 12.2 | 28.6 | 59.2 |

4.4. Results: nadir pressures

The identification of nadir pressure points (equation 5) is a major information need for turbine designers and operators, specially when the target species are notoriously sensitive to barotrauma (e.g. salmonid species). Other studies have shown three patterns manifested in our present results as well. First, nadir pressure points (figure 6a) are for the most part found in the suction side of the runner blades [7, 8]. The large particle sample allowed us to obtain frequency distributions of nadir pressure for each operating point. With these distributions, percentile values (1%, 25%, 50%, 75% and 90%) were computed and plotted in figure 6b. Second, lower discharges tend to yield safer pressure environments for fish [5, 8], which is also indicated by the higher values of pressure in operating point D (figure 6b). Third, the calculation of nadir pressures is overall insensitive to the trajectory modeling approach, i.e., streamlines, Lagrangian spheres and other advanced particle models yield nearly the same nadir pressure distributions [19].

![Figure 6](image.png)

5. Closing comments

In conclusion, the modeling approach introduced herein can represent the potential trajectory of fish passing through turbine flows better than streamlines alone do. Such statement is based
on the following arguments: (i) the collisions are resolved and not modeled, (ii) surface and body forces are accounted for in particle tracking formulations, and play a major role in the particle trajectory, (iii) particle-based modeling data can be directly compared to collision data collected with sensor fish in prototypes and observational data with beads in physical models. The method was developed with some assumptions and limitations that need to be re-stated in order for the reader to put the presented results in context. First, this approach is intended for comparative studies; that is, we do not claim that the method can quantify mortality of fish through turbines in operation. Second, we applied turbulence modeling strategies typically followed in industry practices (URANS), which in consequence introduced lower particle tracking resolution in comparison to what detached- and large eddy simulations would do. However, such advanced turbulence modeling techniques may preclude the implementation of particle-based evaluations in the turbine design process. And third, the inclusion of biological response data for target fish is a necessary step when comparing results to field data with live fish, a practice that has been conventionally favored in fish friendliness studies onsite.

We are currently conducting the validation of the presented modeling approach with the use of bead data collected in simplified geometries of stay and guide vanes in a hydraulic flume. This is offering us the opportunity to revisit the modeling assumptions and add considerable certainty to the estimates of collision rates and nadir pressure with Lagrangian particles. In addition, we are at present revisiting the same analysis with discrete cylindrical and spherical particles that account for particle surface contacts with walls (instead of particle centroids as in this study). We expect to introduce a summary of these validation tests as part of the conference presentation.

References

[1] Coutant C C and Whitney R R 2000 Transactions of the American Fisheries Society 129 351–380
[2] Hogan T W, Cada G F and Amaral S V 2014 Fisheries 39 164–172
[3] Pracheil B M, DeRolph C R, Schramm M P and Bevelhimer M S 2016 Reviews in fish biology and fisheries 26 153–167
[4] Deng Z, Carlson T J, Richmond M C et al. 2007 Sensors 7 3399–3415
[5] Dauble D D, Deng Z, Richmond M C, Moursund R A, Carlson T J, Rakowski C L and Duncan J P 2007 Biological assessment of the advanced turbine design at wanapum dam, 2005 Tech. rep. Pacific Northwest National Laboratory (PNNL), Richland, WA (US)
[6] Ventikos Y, Sotiropoulos F, Fisher Jr R, March P and Hopping P 1999 Waterpower 99: Hydro’s Future: Technology, Markets, and Policy pp 1–10
[7] Richmond M C, Serkowski J A, Ebner L L, Sick M, Brown R S and Carlson T J 2014 Fisheries Research 154 152–164
[8] Richmond M C, Romero-Gomez P, Serkowski J A, Rakowski C L and Graf M J 2015 Comparative study of barotrauma risk during fish passage through kaplan turbines Tech. rep. Pacific Northwest National Lab.(PNNL), Richland, WA (United States)
[9] Romero-Gomez P and Richmond M C 2014 Renewable Energy 71 401–413
[10] Romero-Gomez P and Richmond M C 2014 GMREC, Marine Energy Technology Symposium, Seattle, WA. (April 2014)
[11] Romero-Gomez P and Richmond M C 2017 Journal of Hydraulic Research 55 706–720
[12] Richmond M C, Serkowski J A, Rakowski C, Strickler B, Weisebeck M and Dotson C 2014 Hydro Review 33 88–97
[13] Siemens 2017 Star-ccm+ user guide 12.06 Tech. rep. Siemens PLM Software
[14] Padhy M K and Saini R 2008 Renewable and Sustainable Energy Reviews 12 1974–1987
[15] El-Batsh H and Haselbacher H 2001 Modeling particle deposition on compressor and turbine blade surfaces (na)
[16] Siemens 2011 Computers & Structures 89 2093–2112
[17] Saffman P 1965 Journal of fluid mechanics 22 385–400
[18] Gosman A and Loannides E 1983 Journal of energy 7 482–490
[19] Richmond M C and Romero-Gomez P 2014 IOP Conference Series: Earth and Environmental Science vol 22 (IOP Publishing) p 062010
[20] Deng Z, Carlson T J, Dauble D D and Ploskey G R 2011 Energies 4 57–67