Application of Biological Design Criteria and Computational Fluid Dynamics to Investigate Fish Survival in Kaplan Turbines

By Laura A. Garrison and Richard K. Fisher, Jr, Voith Siemens Hydro Power Generation, Inc.; Michael J. Sale and Glenn Cada, Oak Ridge National Laboratory, USA

ABSTRACT:

One of the contributing factors to fish injury in a turbine environment is shear stress. This paper presents the use of computational fluid dynamics (CFD) to display and quantify areas of elevated shear stress in the Wanapum Kaplan turbine operating at four different flow conditions over its operating range. CFD observations will be compared to field test observations at the same four flow conditions. Methods developed here could be used to facilitate the design of turbines and related water passages with lower risks of fish injury.

BACKGROUND:

In 1995 the U.S. Department of Energy (DOE) initiated the Advanced Hydropower Turbine Systems program (AHTS) to develop design concepts for improving the compatibility of hydro turbines with the environment. One of the recommendations coming from the first phase of the work was that the DOE could contribute to improving the fish-friendliness of turbine designs by providing biological design criteria to guide hydro turbine designers. One of the potential injury mechanisms contributing to fish mortality in Kaplan turbines is shear stress (Ref. 1, chapter 4). Shear can occur where the water interacts with the mechanical structure of the turbine and also where secondary flows are present in the mean flow, such as those characterized by vortices (Ref. 2). In 1999, DOE began a study to define the biological response of fish to the levels of shear experienced inside turbines. The project utilized a special test facility constructed by Pacific Northwest National Laboratory (PNNL) using a jet flow to simulate a shear environment (Ref. 3). That study provided a quantitative description of the intensity of the shear stresses arising from the test jet and quantified the relationship of the stresses to direct and indirect biological effects on fish.

The next step in developing design criteria for shear stress was to determine the areas of high shear in a turbine environment and the source of high shear for each region (Ref. 4). The Wanapum plant on the Columbia River in Grant County, Washington, was chosen for the first phase of this study because it is located in a region where concerns for safe fish passage are significant and because balloon-tagged fish survival testing had been carried out at the plant (Ref. 5).

Figure 1 shows a cross section of a typical Kaplan similar to the Wanapum facility. The CFD work represented in this paper covered all parts in contact with the water from the intake, through the runner and out through the draft tube.

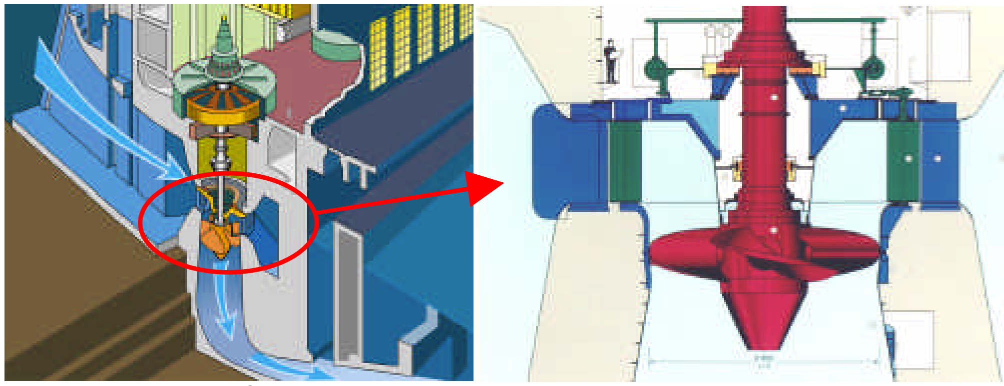

Figure 1: A cross section of a Kaplan turbine showing the entire turbine and its water passages on the left and details of the stay vanes, wicket gates and runner with hub and nearby water passages on the right. The water enters through the semi-spiral case, passes through the stay vanes and wicket gates, then through the runner, and finally into the draft tube, exiting to the tail waters.

METHODS

The Wanapum flow channel was broken down into three sections for the CFD analysis: 1) the intake region including semi-spiral case, stay vanes and wicket gates, 2) the runner region, and 3) the draft tube region. Calculations were completed using the 3-D viscous flow solver, TASCflow, with Reynolds-Averaged Navier-Stokes equations and a Kato-Launder k-epsilon turbulence generation model (Ref. 6). A separate calculation for the stay vane and wicket gate region was used to generate an inflow boundary condition for the runner region and the results of the runner calculation were used to provide an inflow condition for the draft tube. Getting reliable CFD results in a turbine environment can be extremely challenging. Results depend on the grid density and quality as well as selections of a myriad of boundary condition options, turbulence models and computational parameters. Without close attention to the fine details of the problem and a significant background using CFD and performing physical experiments, it is easy to get unreliable and inconsistent results. The CFD methods can only be validated if comparisons such as lab or field data are available. The methods used for the current calculations have been developed with over ten years of 3-D Navier-Stokes CFD using laboratory correlation. However, no field or physical model shear data were available for the Wanapum turbine.

Post-processing was completed using Ensight software version 7.4.0. Shear stress for each calculated region was computed by using a maximum shear stress variable available in Ensight. This calculation approximates the maximum shear stress in any direction at each node within the computational domain (Ref. 7). Previously, strain rate was used as an approximation of shear stress (Ref. 4). For the work presented in this paper, shear stress was used directly in order to get a more accurate view of areas that may injure fish.

Correlation with PNNL Jet Testing

In order to validate the shear stress calculations, CFD predictions were compared to velocity measurements of the jet in the PNNL flume. Figure 2 is a contour plot comparison of the velocity field between the CFD calculations and the pitot probe measurements for the operating condition in which the jet has an an average velocity of 60 fps leaving the nozzle. Figure 3 is a graphical comparison of the CFD velocities in the flow direction compared to the PNNL Laser Doppler Velocimetry (LDV) measurements at several locations (see figure 4 for the locations). A shear stress comparison at the same operating condition can be found in figure 5.

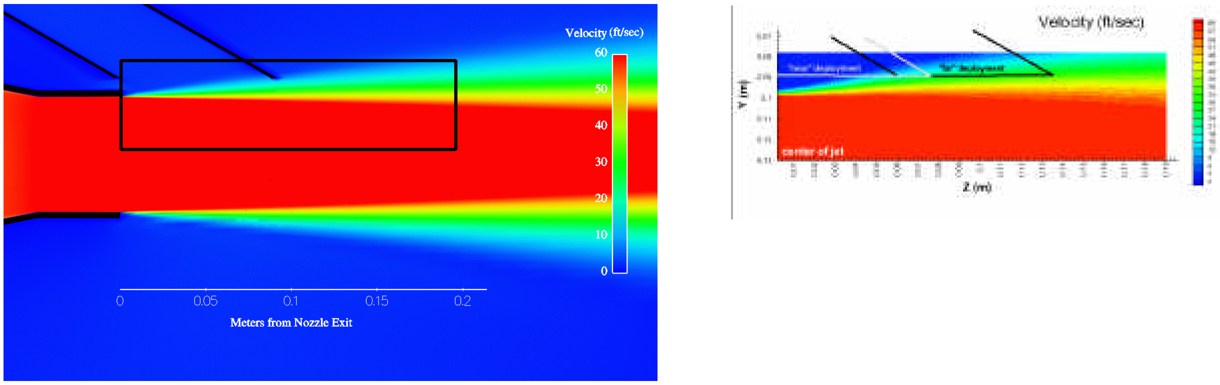

Figure 2: Contour plots of the velocity in the flow direction as calculated with CFD (left) and as measured with pitot probes at the PNNL flume (right) (Ref. 3). The nozzle was operating with an average nozzle exit velocity of 60 fps.

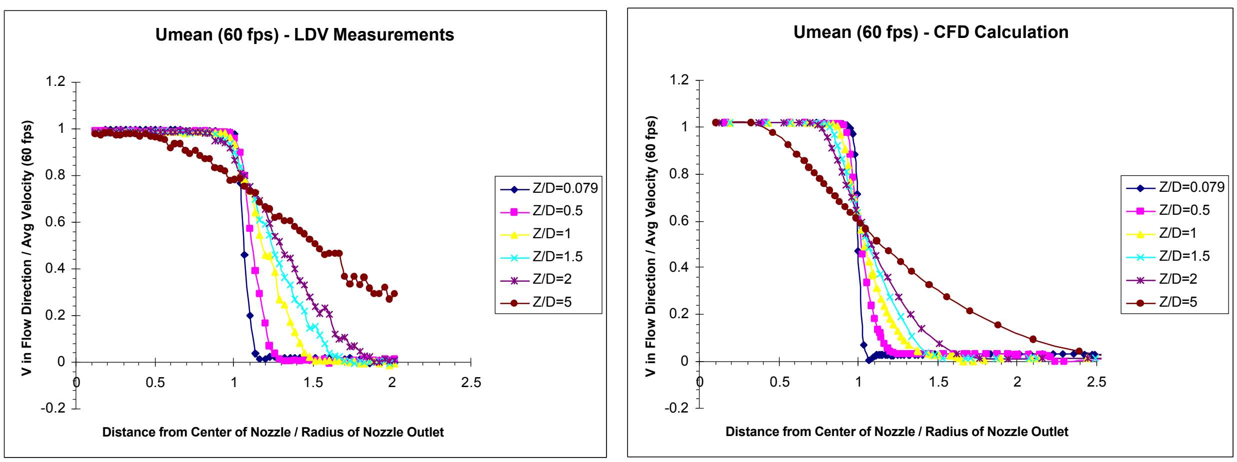

Figure 3: Graphs of the velocity in the flow direction as measured with Laser Doppler Velocimetry (LDV) at the PNNL flume (Ref. 8) (left), and as calculated with CFD (right).The nozzle was operating with an average nozzle exit velocity of 60 fps.

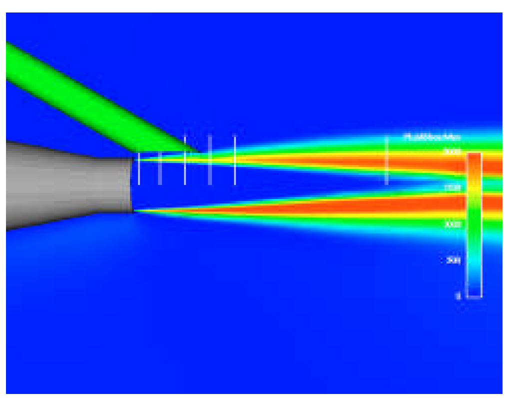

Figure 4: z/d (distance away from the nozzle exit divided by the diameter of the nozzle exit) for the CFD data comparison shown in figures 3 and 5. The white line closest to the nozzle exit is the z/d=0.079 position and the line farthest from the nozzle exit is the z/d=5 position (5 nozzle diameters downstream of the nozzle exit).

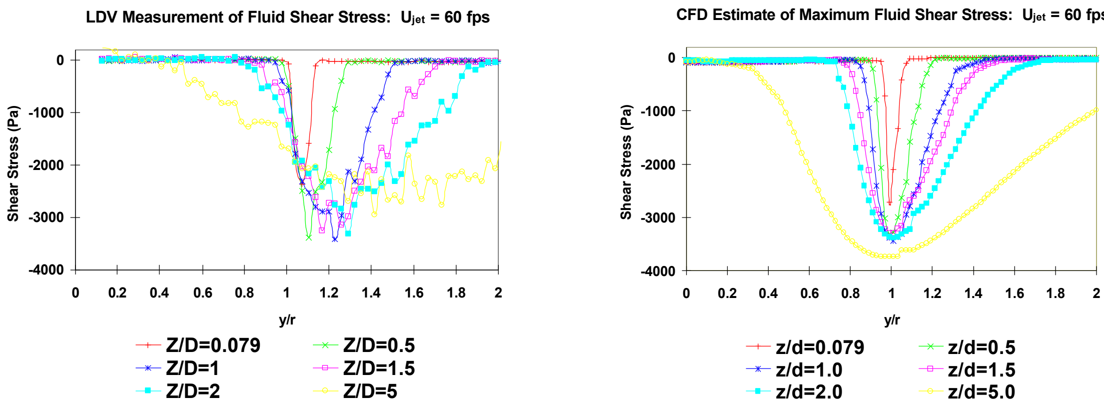

Figure 5: Turbulent shear stress as calculated from the LDV measurements in the dominant direction at PNNL (Ref. 8) (left) and maximum shear stresses in all directions calculated using a CFD (right) approximation. In the legends and axis titles above, z is distance in the flow direction, y is the vertical direction, r is the radius of the nozzle, and d is the diameter of the nozzle. Thus y/r = 1 is at at the same elevation as the wall of the nozzle and z/d = 1 is one diameter downstream of the nozzle exit.

The comparison between CFD and measured data are relatively good, especially in predicting the maximum shear near the edge of the jet for the z/d positions = 0.5 through 2. The CFD calculations over-predicted the shear somewhat in the far field. Part of this discrepancy may be due to the difficulty in both measuring using LDV and predicting using CFD of such a non-uniform, unsteady flow in the far field. Considering the non-exact relationship between shear and fish injury, the shear comparison should be well within the tolerances of this study. Note that these methods are an improvement over those used in Ref. 4 in which strain was used as an estimate of shear stress.

RESULTS OF THE WANAPUM CFD CALCULATIONS

To find the areas where the fish could be injured due to high shear, a threshold level of shear stress was determined based on the flume tests at PNNL. Since the first minor fish injuries for the weakest fish occurred with the flume operating at 30 fps (Ref. 3), a CFD calculation of the flume was completed at 30 fps and the maximum shear stress from that calculation used as the threshold value for possible fish injuries. This calculated threshold value was 1600 Pa.

Four operating conditions (flow rates) for the existing Wanapum turbine were calculated: 9000 cfs, 11,000 cfs, 15,000 cfs, and 17,000 cfs, all at 75 ft. net head. These conditions were chosen to match fish survival-testing conditions previously completed at the Wanapum power plant. The greatest areas of shear stress in intake and draft tube occurred at the highest flow condition, 17,000 cfs, while the highest shear in the runner occurred at the lowest flow condition, 9,000 cfs . Figures 6-9 illustrate areas of shear above the threshold of 1600 Pa.

Turbine Intake Region

There were no large areas of high shear stress in the intake region (see figure 6) upstream of the stay vanes at any of the four conditions. Some shear above the threshold value of 1600 Pa was found in some of the wicket gate wakes at all operating conditions. The higher flow conditions had some high shear at the entrance edge of one or two stay vanes and a small amount on the band at the point of highest curvature. The high shear areas in the wicket gate wakes are due to a combination of band curvature with overhanging gates and differences in

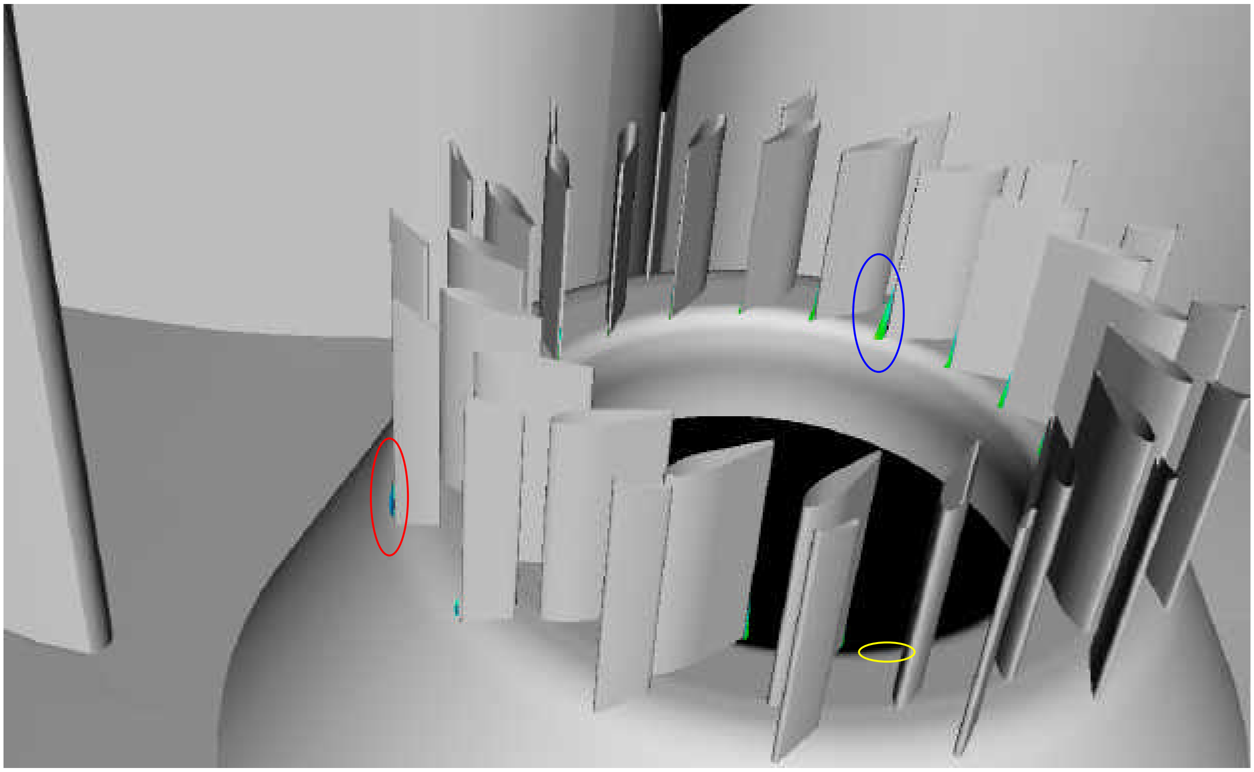

Figure 6: Areas in the intake region with shear above the fish-injury threshold of 1600 Pa when the turbine is operating at 17,000 cfs. These areas are represented by isosurfaces of constant shear stress = 1600 Pa. They are colored with speed to increase their visibility. The type of high shear region enclosed with the red ellipse is due to a poor incidence angle on the stay vane entrance edge. Many of the wicket gates had high shear in their wakes (blue ellipse). There was also a very small region of high shear on the band (yellow ellipse).

velocity from one side of the gate to the other. The stay vane entrance edge shear is due to poor flow incidence angles. The band shear is caused by the velocity change as the water accelerates around the band curvature.

Turbine Runner Region

Shear above the threshold value of 1600 Pa occurred in four main areas in the runner section: 1) in the boundary layer on the blade surface, 2) at or near the periphery gap, 3) at or near the hub gap, and 4) in the wake under the blade. Figure 7 shows the shear at or above threshold for the 17,000 cfs case with no hub gap and the 11,000 cfs case which has both hub and periphery gaps. The high shear regions near the gaps are caused by “squeezed” flow through the gaps and resulting vortices (figure 8).

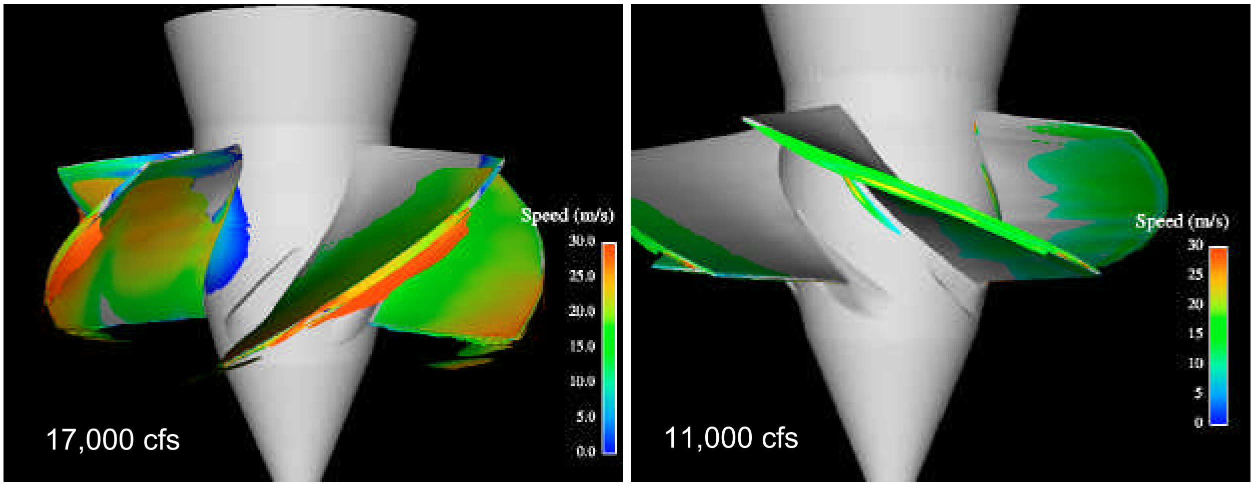

Figure 7: Areas in the runner region with shear stress above the fish-injury threshold of 1600 Pa at 17,000 cfs (left), and 11,000 cfs (right). The high shear areas are represented by isoSurfaces of shear stress = 1600 Pa and are colored with speed. Any non-gray area is above the threshold.

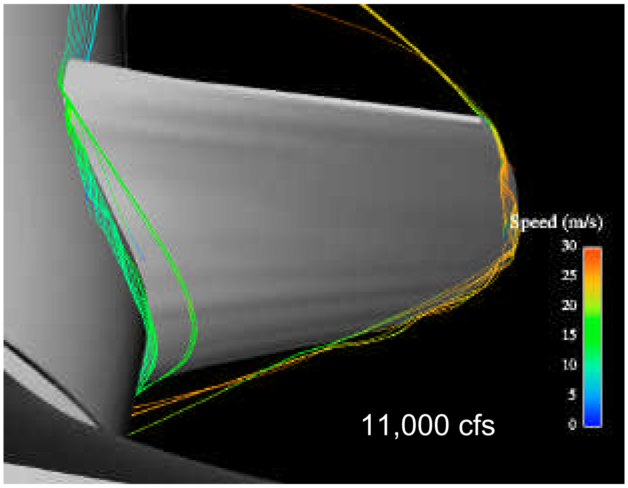

Figure 8: Streamlines at the hub and periphery gaps show vortices that lead to high shear stress.

Turbine Draft Tube Region

The high shear regions in the draft tube resulted more from the runner geometry than the draft tube design. The water passage area under the hub (which is part of the runner) contains a vortex that, at higher flow rates, causes shear above the threshold. This vortex, along with the flow angles as the water hits the piers, caused high shear regions at the piers. Figure 9 illustrates the areas of high shear at the highest flow rate, 17,000 cfs.

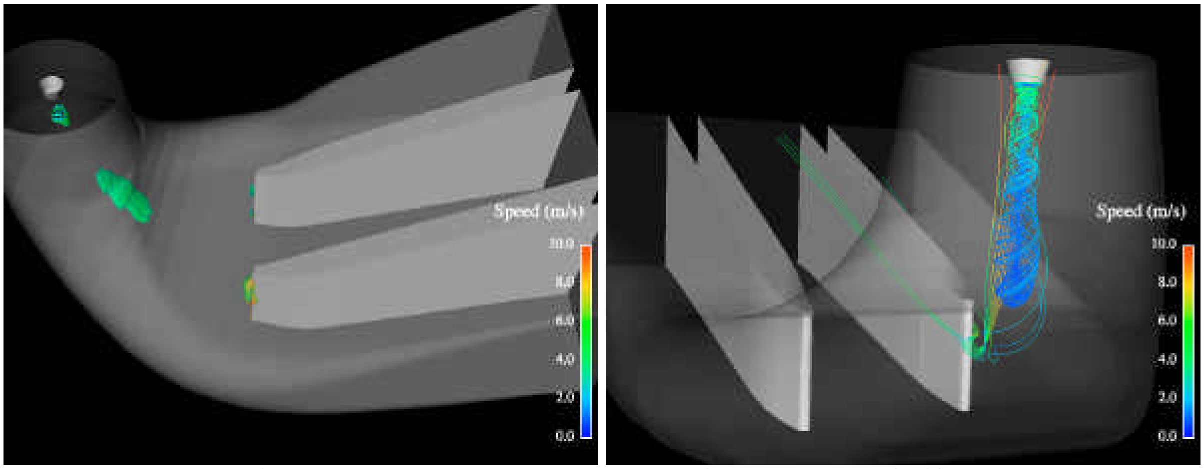

Figure 9: Areas of shear above 1600 Pa in the draft tube region (left). The high shear areas are represented by isoSurfaces of shear stress = 1600 Pa and are colored with speed. Streamlines showing the "torch" vortex under the hub which causes the high shear in the upstream portion of the draft tube (right). Streamlines also illustrate the poor incidence angle at the pier due to the torch vortex and uneven flow from the runner.

COMPARISON OF OPERATING CONDITIONS AND COMPONENTS

Plots of isoSurfaces and streamlines help to visualize the areas of high shear and to determine how to redesign to lessen or eliminate these regions. However, it would be helpful to be able to quantify the high shear within a particular turbine component in order to do a direct comparison between designs and operating conditions. The best criterion would be the probability that any fish entering the intake passes through a high shear region. Since we do not have sufficient fish trajectory data, estimates can be made using percent of cross-sectional area in high shear regions and percent of flow through high shear regions at various cross-sections. The percentage of cross-sectional area assumes that a fish has an equal likelihood of being at any position. The percentage of flow at each cross-section further refines this estimate by assuming more fish will pass through high flow areas than low flow areas.

Percent of Cross-Sectional Area Where Shear > Threshold

Figure 10 is a cross-section of the draft tube just under the hub; the area colored with red is above threshold shear stress and blue is below threshold. These types of slices (cuts across the machine oriented perpendicular to the main flow direction) were used throughout the machine to develop the graphs in figures 11 and 12. Figure 11 is a plot of the ratio of cross-sectional area that contains shear stress above the threshold value from just upstream of the stay vanes to the draft tube exit. Note that the worst case for the runner is the low flow condition, 9000 cfs, while the high flow condition, 17,000 cfs, causes high shear from the hub vortex and draft tube piers.

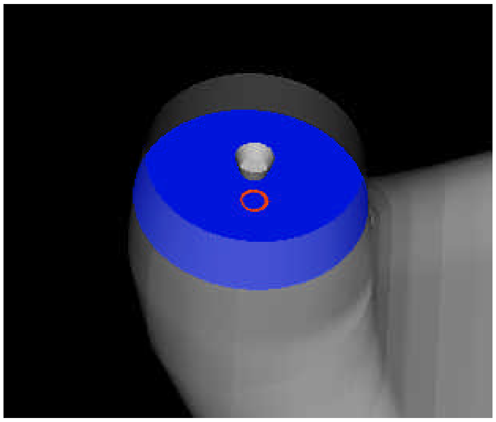

Figure 10: A cross-section of the draft tube just under the runner hub. The red area is greater than the shear stress threshold of 1600 Pa and the blue is less than 1600 Pa.

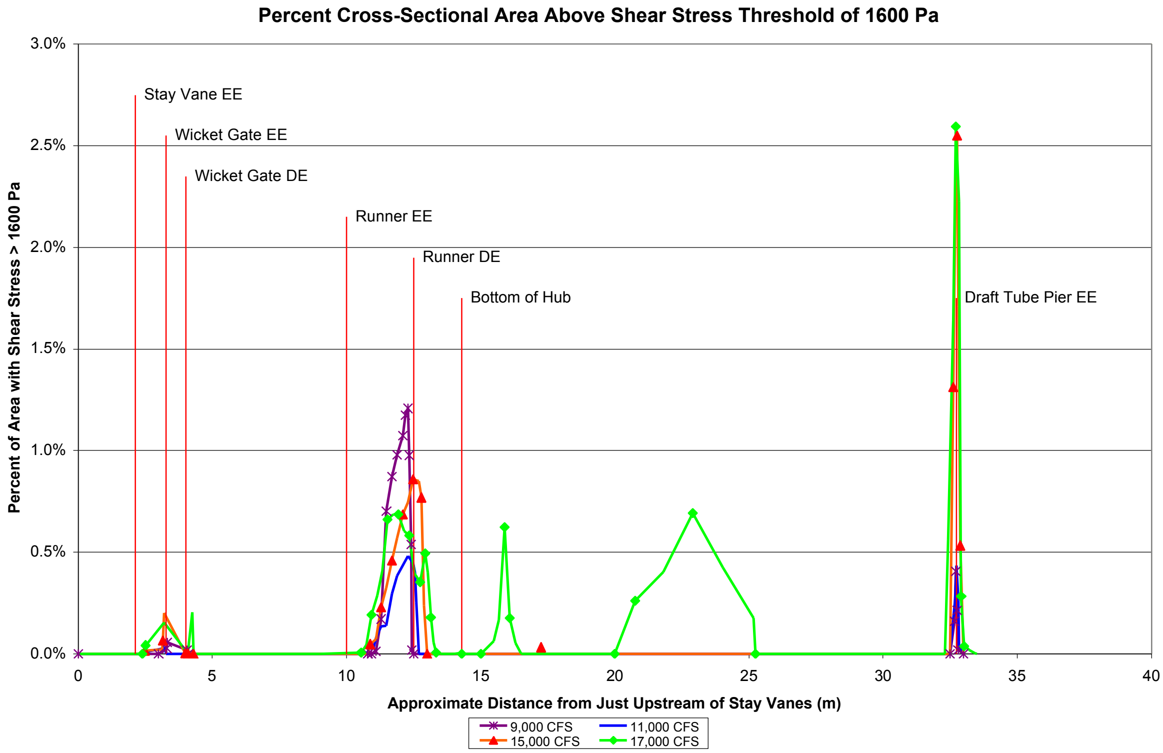

Figure 11: Comparison of operating conditions starting just upstream of the stay vanes. EE stands for Entrance Edge and DE is Discharge Edge. At the higher flow rates, the peak area of high shear (2.5%) occurs at the nose of one of the piers. For the runner region, the worst operating condition for shear is at low flow, 9000 cfs. This includes the shear stress within the boundary layer. All stay vanes, wicket gates, and runner blades are included in the graph.

Percent of Flow Through Cross-Sectional Area Where Shear > Threshold

Figure 12 is similar to figure 11, except the values have been normalized with flow rate. This assumes that more fish will pass through high flow areas than low flow areas. The plots are similar, but with lower values meaning the high strain and turbulence that cause the high shear tend to occur where the flow is less than average, for example near stagnation zones, areas of backflow, and boundary layers.

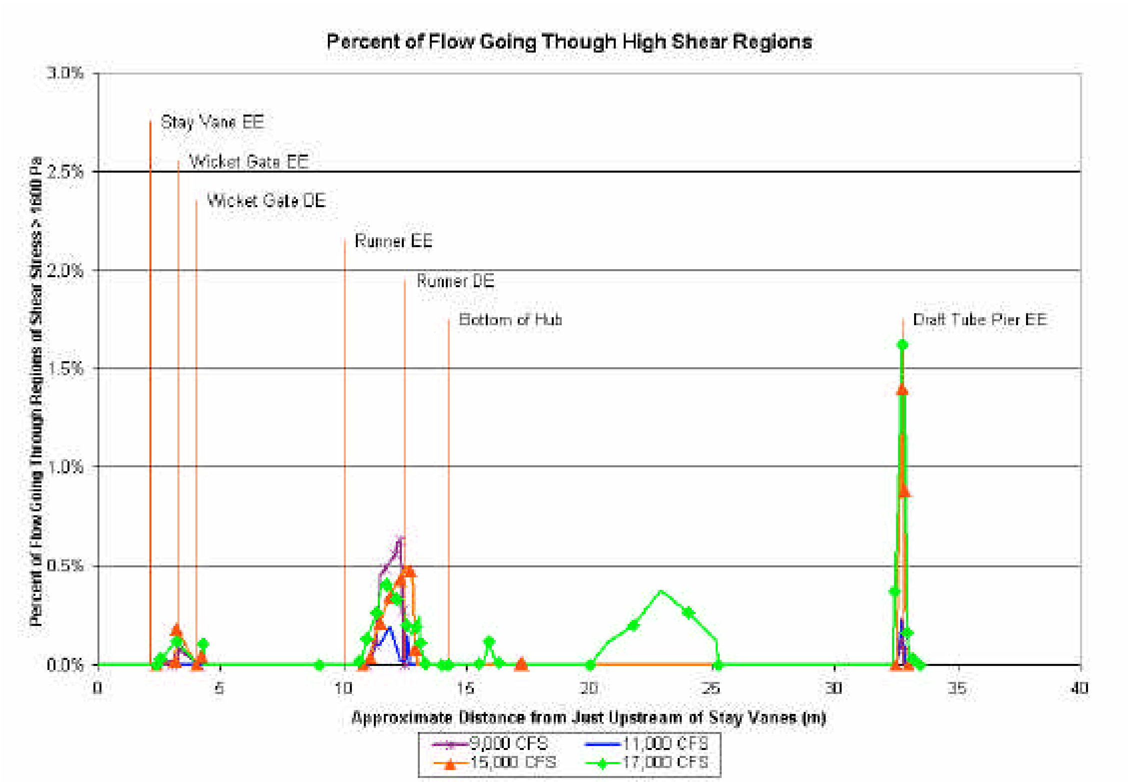

Figure 12: Comparison of operating conditions starting just upstream of the stay vanes. EE stands for Entrance Edge and DE is Discharge Edge. This is similar to the area ratios in figure 11, with the percentage values somewhat lower.

COMPARISON WITH FIELD TESTED FISH SURVIVAL DATA

Figures 13 and 14 show the field-tested survivability of fish passed through the Wanapum turbine using balloon-tag methods (Refs. 1 and 5). The colored area above the survival line represents the mortality of the fish, and has been broken down into suspected causes. Figures 13 and 14 contain survival data for fish released near the centerline of the intake, and near the roof of the intake, respectively. The shear stress as presented in this report would represent categories from figures 13 and 14 of avoidable losses, deflector (bottom of the hub in the draft tube cone) related turbulence, and hub and gap related turbulence. Figure 15 shows an estimate of the shear stress mortality plotted on top of the average survival from figures 13 and 14. The shear mortality estimate was calculated by multiplying one minus the peaks in figure 12. In other words, if 100 fish enter the turbine operating at 17,000 cfs, about 99.8% would presumably safely pass through the stay vane and wicket gate area (see figure 1 for a diagram of these components). These 99.8 fish would then be subject to the runner where about 99.8% x 99.6% = 99.4% would safely pass. This continues through each section. The results from this estimate show an increasing shear-related mortality with flow rate. This does not compare well with the fish survival data, indicating that the fish are probably not evenly distributed throughout the flow channel or distributed according to flow rate only. For example, the fish may be swept around the draft tube pier before ever reaching the high shear region (see figure 9). It is also possible that fish are drawn into the gap areas by low pressure caused by jet flow through the gaps. An estimate of the trajectories that a fish will likely take is a missing piece of this puzzle. The data also suggest that factors besides shear stress may be playing a large part in fish mortality.

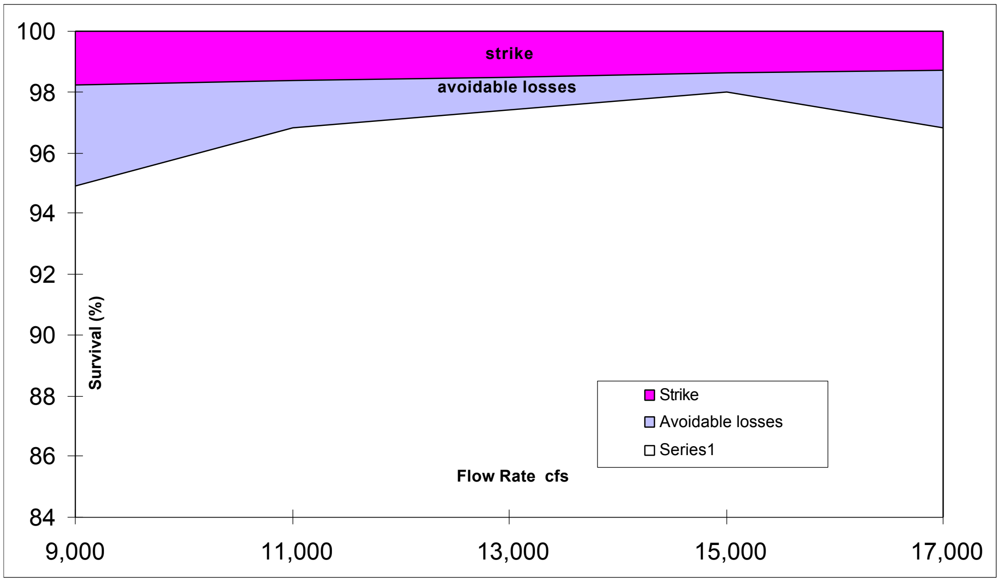

Figure 13: Estimates of survival for fish released near the centerline of the unit with estimated causes of mortality (Ref. 1). It is assumed that the fish passed near the center of the blade, thus avoiding any gap-related injuries and any draft tube vortex-related injuries.

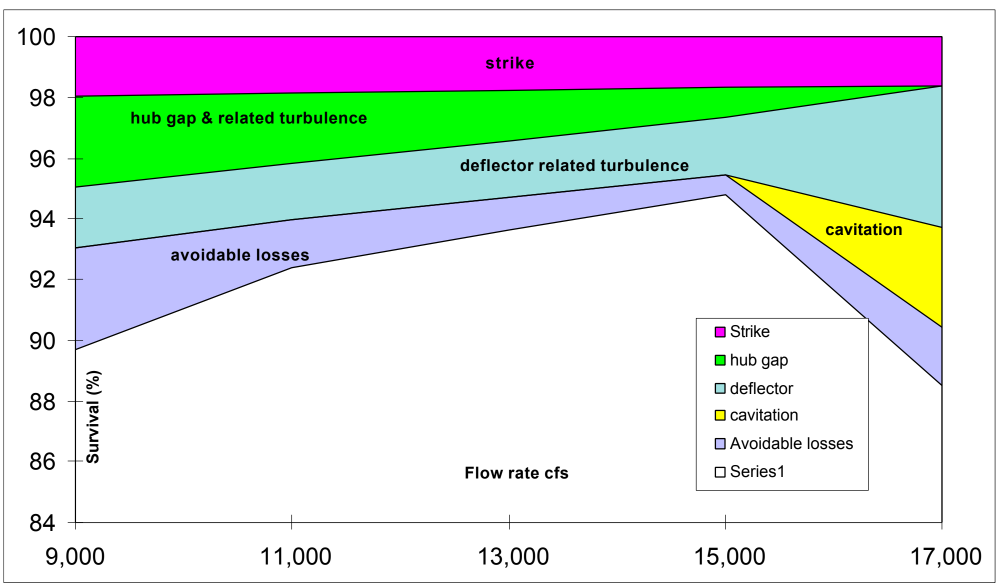

Figure 14: Estimates of survival for fish released near the hub with estimated causes of mortality (Ref. 1). It is assumed that the fish passed near the hub and were subject to any harmful effects of the hub gap as well as those from the draft tube vortex.

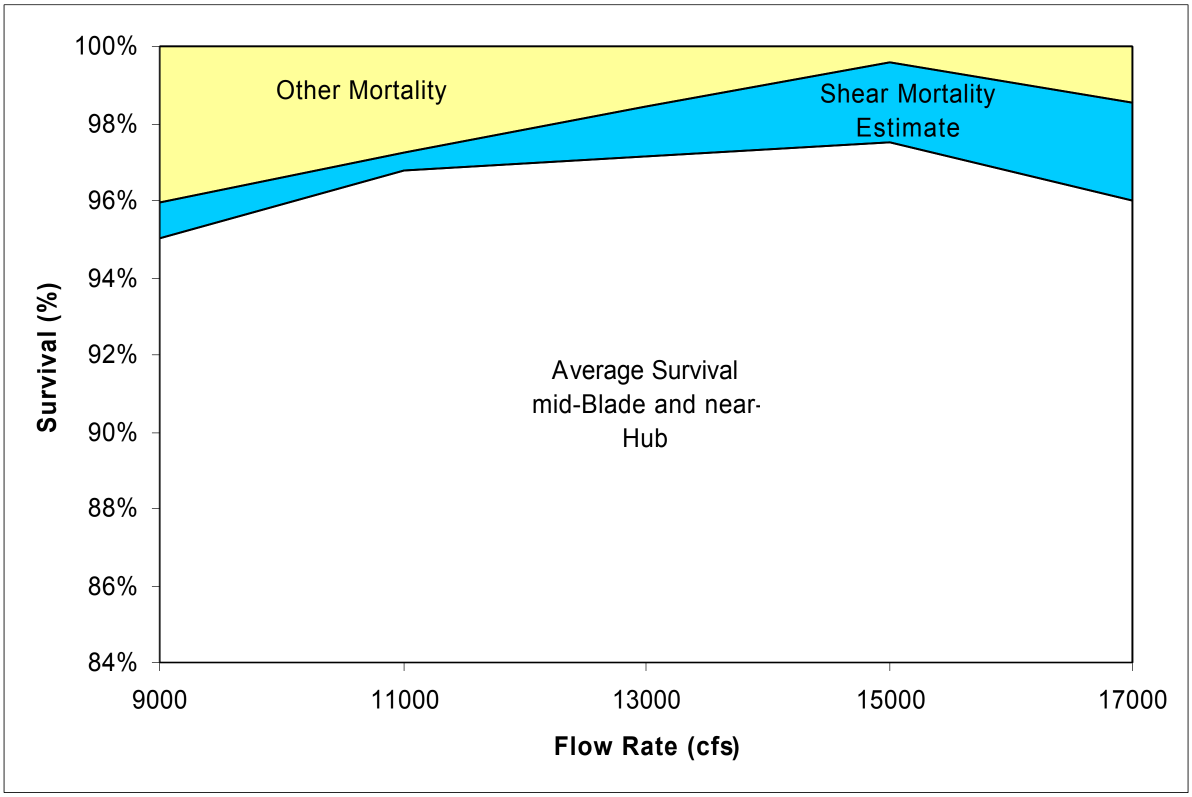

Figure 15: CFD estimate of mortality through the entire turbine water passage due to calculated shear stress (cyan). The calculated shear stresses are not at a minimum at the 15,000 cfs point as would be expected from the average survival data (white). The survival data are an average of the fish release data from figures 13 and 14. For the calculations, the minimum shear area occurs at the peak efficiency operating condition, 11,000 cfs. This suggests that probable fish trajectories need to be taken into account when calculating the mortality due to shear.

CONCLUSIONS:

The work presented in this paper demonstrates how the combination of biological design criteria and CFD can be used to understand what is happening inside hydropower turbines. This approach provides valuable estimates of locations and sizes of the high shear regions within a turbine environment. The methods can be used to compare various designs in terms of the areas and volumes of high shear regions, thus aiding in the design of turbines that cause little or no harm to fish. However, it cannot predict the fish mortality directly. Several pieces of the puzzle are still missing, the most important being: What path do the fish actually take through a turbine? Other phenomena that could contribute to fish mortality should also be studied further. Here are some other questions yet to be answered:

- When in contact with a high shear region, is exposure time important?
- Can a fish get caught in the hub vortex and remain there for a period of time, leaving injured and/or disoriented?
- If cavitation is present, what are its effects?
- How likely is it that fish will enter the dangerous gap and wake regions?
- How can CFD modeling be improved to represent more of the unsteady effects of turbulence?

REFERENCES:

1. Franke, G. F., D. R Webb, R. K. Fisher, D Mathur, P Hopping, P. March, M. Headrick, I. Laczo, Y. Ventikos, and F. Sotiropoulos, "Development of Environmentally Advanced Hydropower Turbine System Design Concepts," York, PA: Report prepared for U.S. Department of Energy, Assistant Secretary for Energy Efficiency and Renewable Energy, DOE Idaho Operations and Hydropower Research Foundation, Inc.
2. Fisher, Brown, and Mathur; "The Importance of the Point of Operation of a Kaplan Turbine on Fish Survivability," presented at Waterpower '97 conference Atlanta, GA, August 1997.
3. Neitzel et al; "Laboratory Studies on the Effects of Shear on Fish: Final Report;" United States Department Of Energy DOE/ID-10822; September 2000. (also see: http://www.inel.gov/national/hydropower/turbine/shearfish10822.pdf/)
4. Fisher, R.K, Garrison, L.A., Brown, S., Sale, M.J., "On the Fish Survival Characteristics Related to Shear Flows in Kaplan Turbines," WaterPower XII 2001, Salt Lake City.
5. Normandeau Associates, Inc., J. R. Skalski, and MCC. 1996. "Fish Survival Investigation Relative to Turbine Rehabilitation at Wanapum Dam, Columbia River, Washington." Report prepared for Grant County Public Utility District, Ephrata, WA.
6. AEA Technology Engineering Software, Ltd., *CFX-TASCflow Computational Fluid Dynamics Software Primer Documentation Version 2.10,* 554 Parkside Drive, Waterloo, Ontario, Canada N2L 5Z4, 2000.
7. Computational Engineering International, Inc., *Ensight User Manual for Version 7.1*, P.O. Box 14306 Research Triangle Park, NC 27709, April 2000.
8. Courtesy of D. Neitzel at Pacific Northwest National Laboratories.

Acknowledgements

This research has been funded, in part, by a subcontract with Oak Ridge National Laboratory, under the U.S. Department of Energy's Hydropower Program in the Office of Wind and Hydropower, Energy Efficiency and Renewable Energy. Additional support was provided by Grant County Public Utility District.

Authors

Dr. Laura A. Garrison leads the Numerical Flow Simulation group for Voith Siemens Hydro Power Generation, Inc. She obtained her Ph.D. in Bioengineering from Penn State University and has ten years experience in the areas of computational fluid dynamics, fluid mechanics and numerical methods.

Richard K. Fisher Jr. is Vice President, Technology for Voith Siemens Hydro Power Generation, Inc., President of Hydro Resource Solutions, LLC and Chairman of the International Association of Hydraulic Research Section on Hydraulic Machinery and Systems. He has 31 years experience in the hydro industry.

Dr. Michael J. Sale is a Distinguished Senior Research Staff Member and Group Leader for Water Resources in the Environmental Sciences Division of Oak Ridge National Laboratory. He has been conducting hydropower related R&D since 1980.

Dr. Glenn F. Cada is a fisheries biologist and Senior R&D Staff Member in the Environmental Sciences Division, Oak Ridge National Laboratory. He has 25 years experience in research and assessment of the impacts of energy technologies on aquatic systems.