APPLICATIONS OF COMPUTATIONAL FLUID DYNAMICS TO PROBLEMS IN WATER RESOURCES ENGINEERING

A. Jacob Odgaard

Iowa Institute of Hydraulic Research, and
Department of Civil and Environmental Engineering
The University of Iowa, Iowa City, Iowa 52242

Introduction

Computational Fluid Dynamics (CFD), the art and science of applying methods of discrete mathematics (numerical analysis) to solution of equations of fluid motion, is rapidly becoming an important tool for analysis and design in many fields, including hydraulics and water resources engineering.

In the Hydraulics and Water Resources Engineering Program at the Iowa Institute of Hydraulic Research, CFD is used for calculating flows in reservoirs and rivers and in and around hydraulic structures. Examples include calculations to support design of fish bypass systems in the Pacific Northwest, development of the next generation of hydraulic machines (pumps and turbines), and development of designs and sediment management strategies at water intakes to eliminate sediment ingestion. In these studies, CFD is used as an adjunct to laboratory experiments and, in some cases, field observations.

The following is a discussion of technological advances and solution strategies that have been accomplished as a result of simulations with CFD models. Examples include sediment management strategies that incorporate the use of submerged vanes. Applications of the submerged vane technique are described.

The paper concludes with reflections on future developments in CFD and the role of CFD in water resources planning and management.

Fish Diversion at Hydroelectric Power Plants

A comprehensive research program is underway at IIHR to develop and test fish bypass systems for hydroelectric powerplants. The program is driven by the urgent need to stabilize, if not restore, the salmon population in the Columbia and Snake Rivers in the Pacific Northwest. The Columbia and Snake Rivers are controlled to a large extent by dams which provide reservoirs not only for hydroelectric power generation but also for irrigation and flood control. There is a concern that the dams may have contributed to the decline in the fish population. Consequently, efforts are underway to make the dams and their appurtenances more fish friendly by building facilities that divert the fish around areas where they may potentially be damaged. One such area is the hydroelectric turbine.

A critical component in the effort is the development of a good understanding of the flow patterns in and around the potential diversion structures. This understanding can be obtained by building hydraulic scale models of the proposed structures and measuring the velocity distributions. However, with the rapid advances in computational methods and computing facilities, numerical flow models are becoming a very attractive alternative to hydraulic scale models. Recent developments suggest that numerical models can provide answers to many of the critical questions at a considerably lower cost.

The numerical models solve the Reynolds-averaged Navier Stokes equations in generalized curvilinear coordinates. Advanced turbulence models are used for computing the eddy viscosity in these equations. Reference is made to Meselhe and Odgaard (1997) and Sinha et al. (1997).

Figure 1 shows a simulation of velocity field and particle paths in the river reach upstream from Wanapum Dam. The reach is seven kilometers long. Three different levels of resolution are obtained by using three different grid sizes. A relatively course grid is used for obtaining the overall velocity distribution (Figure 1a). Near the dam the grid is refined to obtain better resolution of the velocity field (Figure 1b). Very close to the powerhouse and spillway the grid is further refined to permit velocity variations to be described very accurately within and in the immediate vicinity of the intake, spillway, and diversion facilities (Figure 1c). The latter velocity distribution is used for correlation with fish migration data and for detailed design. Figures 2 and 3 show details from the calculations of flow through the turbine and the draft tube downstream from the unit. These calculations are used primarily for efficiency evaluations.

Figures 4 and 5 show results of flow simulations in the river reach downstream from Wanapum Dam. These simulations are used for determining suitable release sites for fish that have been diverted around the turbines. Predation is a major problem in this area, and the fish will have to be released in areas where the chance of predation is minimum.

Sediment Management at Water Intakes

Sediment management is another major focus area of IIHR. Most recently, efforts have been concentrated on sediment management at water intakes. The effort is in response to a documented need for more reliable, quantitative design methods and good methods for predicting performance.

The principal features that influence patterns of sediment movement are depicted in Figure 6 (Neary and Odgaard 1993). An influential feature is the non-uniform vertical distribution of streamwise velocity in the main channel; velocities are low near the bed and higher near the surface. This feature causes the amount of main channel flow captured by the diversion to be greater near the bed, where bed-load transport occurs and where suspended-sediment concentration is highest. As the flow turns to enter the diversion, a secondary vortex forms that convects sediment towards the inside of the diversion where flow has separated. The separated flow has a low velocity that enables sediment to settle and block the diversion. Flow that turns to enter the diversion, but continues down the main channel, forms a secondary vortes in the main channel as well. This vortex may locally erode the main channel bank and the bed. If unchecked, this vortex may substantially alter the diversion entrance.

The numerical model is used for calculating the b[illegible] shear stresses. It is the magnitude and direction of the bed shear stresses that dete[illegible]ine the fate of the sediment that is transported with the flow at and near the bed. [illegible]gure 7 shows calculations of bed shear stress distribution in and around an intak[illegible]or different flow conditions. These bed shear stress calculations are used for determi[illegible]ng sediment management strategies. Reference is made to Neary et al. (1997).

A series of numerical and experimental test were conducted to determine the most cost effective means to minimize both sediment transport into the intake and the volume of sediment accumulation in the intake. A total of 40 different sediment control strategies were tested. It was found that a strategy that employs submerged vanes outside the water intakes in general provides the most desirable results considering cost and feasibility. Figure 8 shows one such strategy. Depending on flow and boundary conditions, submerged vanes would be used either alone or in combination with a skimming wall and/or upstream channel stabilization scheme.

Examples of field installations are given by Wang et al (1996).

Future Developments

The major challenge for the future is the application of CFD models to simulate the fate and transport of constituents in the water environment, such as sediment, heat, dissolved gasses, chemicals, and contaminants. On a broader scale the integration of surface-water and groundwater modelling is inevitable. Ultimately, the models will have to become part of an integrated decision system for the planning and management of our water resources.

References

Meselhe, E.A., and Odgaard, A.J. (1997). "Three-Dimensional Numerical Flow Model for Fish Diversion Studies at Wanapum Dam", *Journal of Hydraulic Engineering*, ASCE, Submitted for possible publication.

Neary, V.S., Sotiropoulos, F., and Odgaard, A.J. (1997). "Three-Dimensional Numerical Model of Lateral-Intake Flows", *Journal of Hydraulic Engineering*, ASCE, Tentatively accepted for publication.

Sinha, S.K., L.J. Weber, and Odgaard, A.J. (1997). "Numerical evaluation of fish bypass outfall location alternatives", *North American Journal of Fisheries Management*, American Fisheries Society, Submitted for possible publication.

Sinha, S.K., Sotiropoulos, F., and Odgaard, A.J. (1997). "Three-Dimensional Numerical Model for Flow through Natural Rivers", *Journal of Hydraulic Engineering*, ASCE, Accepted for publication.

Wang, Y., Odgaard, A.J., Melville, B.W., and Jain, S.C. (1996). "Sediment Control at Water Intakes." *Journal of Hydraulic Engineering*, ASCE, 122(6), pp. 353-356.

Neary, V.S. and Odgaard, A.J. (1993). "Three-Dimensional Flow Structure at Open-Channel Diversions," *Journal of Hydraulic Engineering*, ASCE, 119(11), pp. 1223-1230.

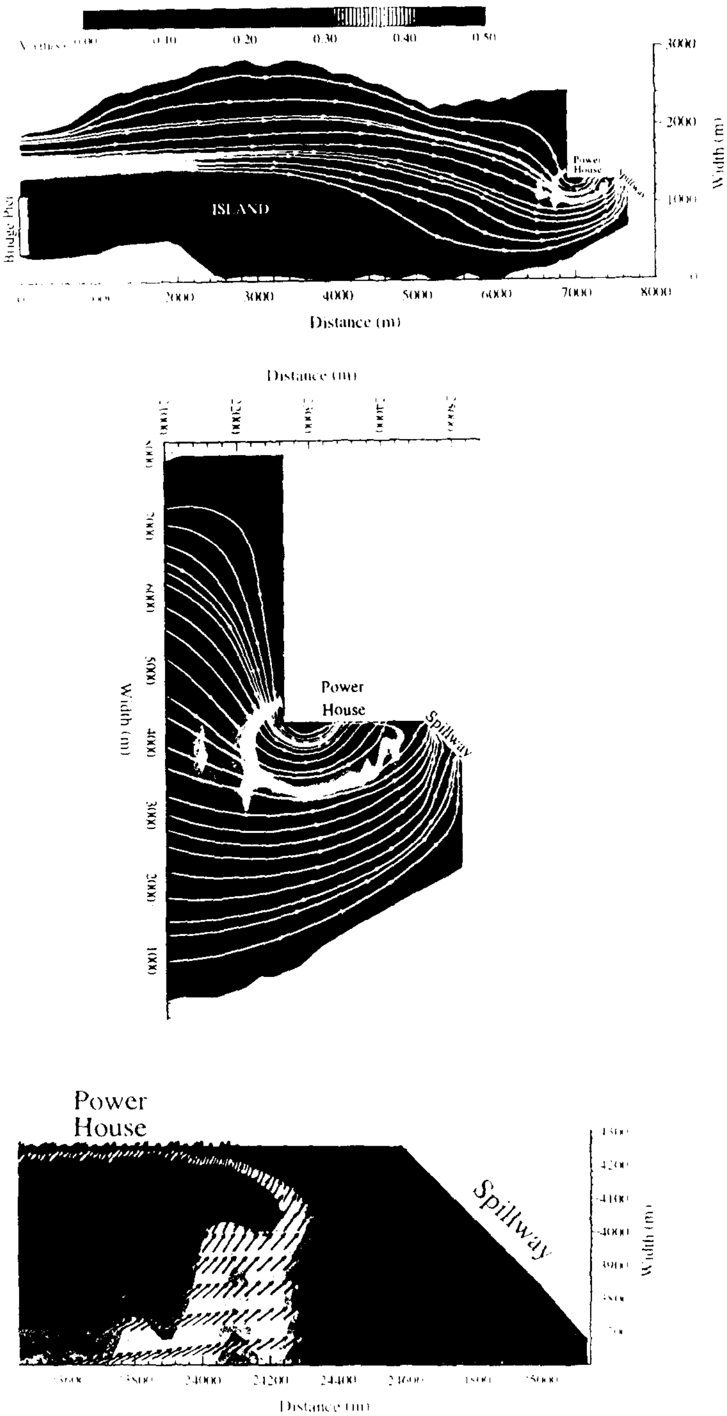

Figure 1. Simulation of velocity field upstream of Wanapum on the Columbia River.

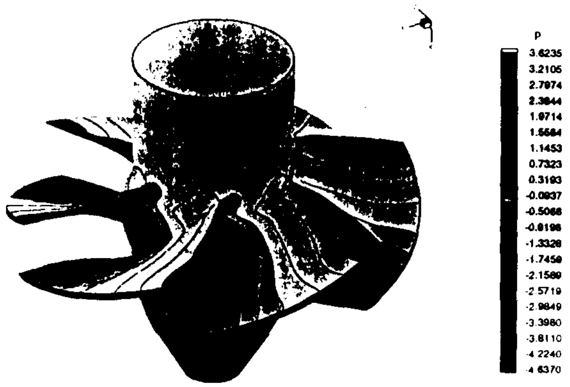

Figure 2. Pressure distribution on turbine hub and blades

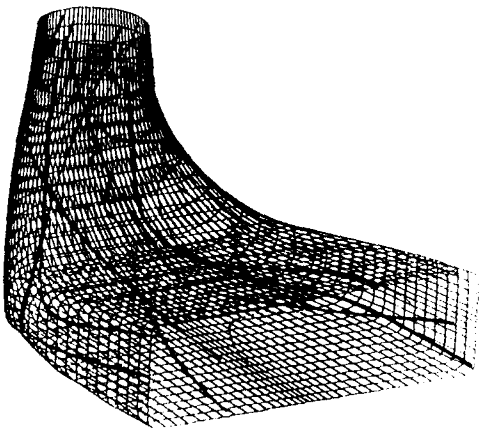

Figure 3. Simulation of flow in draft tube. Particle paths.

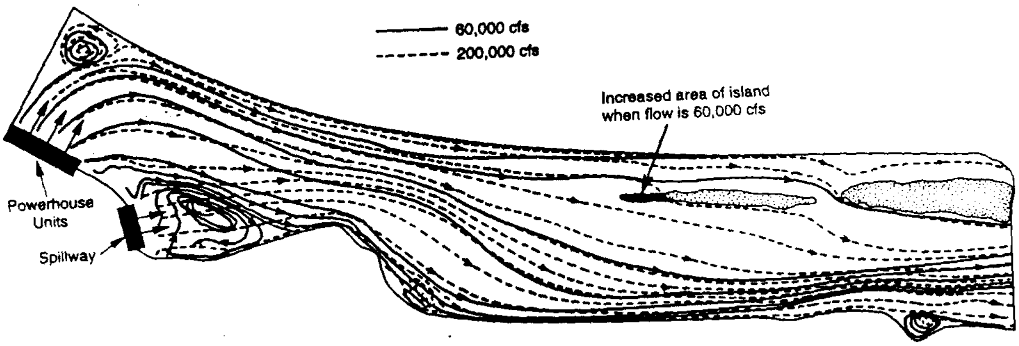

Figure 4. Simulation of velocity field downstream of Wanapum.

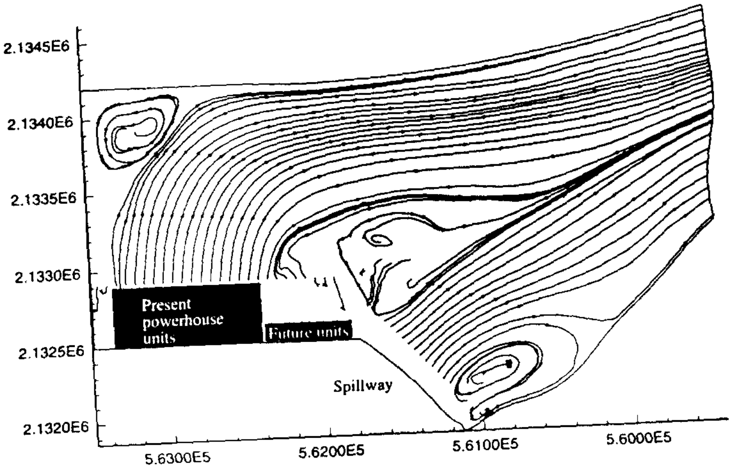

Figure 5. Simulation of velocity field downstream of Wanapum. Details.

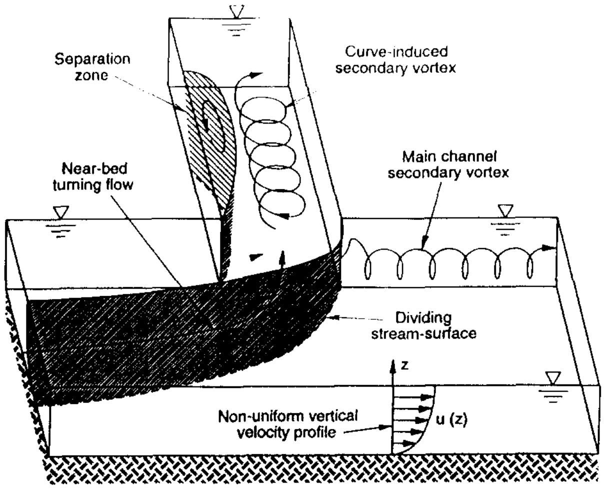

Figure 6. Flow features at diversion

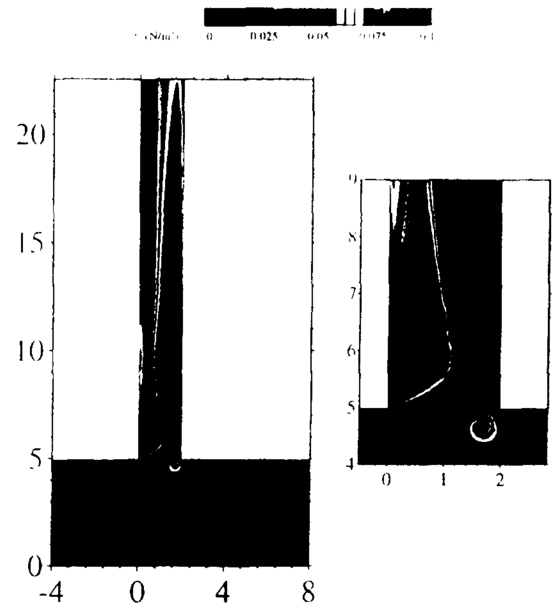

Figure 7. Bed shear stress contours at diversion

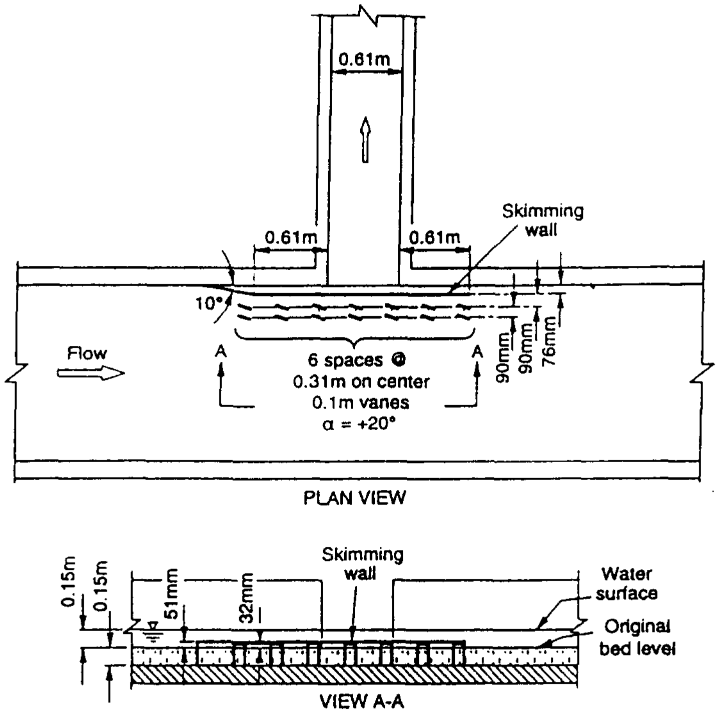

Figure 8. Submerged vanes and skimming wall at diversion