NUMERICAL STUDY OF TURBULENCE AND FLOW VELOCITY IN THE ORIFICE AND WEIR FISHWAY WITH FLUENT
Ebrahim Nohani*

Department of hydraulic Structures, Dezful Branch, Islamic Azad University, Dezful, Iran
*Corresponding author’s Email: nohani_e@yahoo.com

ABSTRACT
An interruption in the flow of rivers such as the construction of the dam is due to the negative impact of dams on promotion of aquatic animal. To solve the environmental problem, a fishway structure is developed that makes the passage of fish through the dams possible. Recent studies have shown that the water velocity and turbulence can have a key role in successful passage of the fish through the fishway. In this study, using Fluent software and some turbulence model, the hydraulic behavior, flow velocity and turbulence in the orifice and weir fishways was evaluated and the results were compared with Instrumentation data. The results showed that turbulence model \(k-\varepsilon\) Standard has more precision in simulation of the flow pattern. Also, the vortex region and safe areas for the passage of fish in the ponds were identified and were classified according to flow rate.

KEYWORDS: Fishway design, Migration of fish, Velocity, orifice and weir, turbulence model, simulation

INTRODUCTION
Design of fishway should be proportional to fish migration, otherwise it will delay the migration of fish and this delay will cause the loss of many of these creatures. It is worth noting that constructing fishway will not solve all biological problems created by a dam in a catchment area, but it only builds a way for fish migration to the upstream. In Iran, the numbers of constructed diversion dams or under construction are very high, but unfortunately this environmental issue (constructing fishway structure) is less considered or if it is considered, these structures do not work properly in some cases. Addressing the components such as data related to velocity distribution and flow pattern, the turbulence is proper by using a numerical model. In such circumstances, the major challenges that can threaten the performance of the structure can be solved.

Identification of fish behavior in selecting the type of fishway is very important, because it has been found that certain kinds of fish prefer a type of fish more than other types. Average speed on the spillway, in the orifice in designing the fishway for mature salmon is considered 2.5 m/s. To present the most appropriate design for the fishway, in addition to recognizing the fishways and how to design them, we should know the kind of fish in order to let fish ascend the designed fishway. In the last half century, the types of fish have been designed and implemented in different parts of the world, that according to Smith,(1985) these structures can be divided into the following types: orifice fishway, weir fishway, orifice and weir fishway, vertical slot fishway, Denil fishway, fishway with the hand-packet aggregates and other local materials. Orifice and weir fishway are formed from consecutive ponds among which the separating walls exist. The wall can be used as a weir in the structure and allows the fish to jump if necessary and migrates from the downstream pool to the upstream. Also under the wall, an outlet may be created that the fish that are not able to jump to use this orifice and go to upstream. Orifices are usually placed as zigzag at the both sides of the middle wall. Depending on the height of dam, the number of pools may be more than 10. Usually every 5 ordinary pool, a rest pool is considered. The fishway is separated from other dam components through the sidewall. It is usually built on one side of the diversion dam close to the river bank. The maximum water flow through the orifice should not exceed fish suicidal speed (the highest achievable speed in less than 15 seconds).

Katopodis and Rajaratnam, (1997) performed much research on flow hydraulic in fishways, especially the Denil fishway and offered useful results to design this type of fishways. Shamlou and Aknourni, (2012) studied the flow in the pond and weir fishway and showed that flow is formed in this type of fishways in both skimming and immersed modes (Fig. 2)
Kim et al. (2011) showed that by analysis of stimulated flow and the specification of flow given the type of fish, the flow can be adjusted in a way that fish can move easily in it. Calluad et al. (2010) proposed the installation of barrier opposite the outlet orifice of fishway with a vertical gap to modify the performance of the structure. Alvarez et al., (2011) to obtain the fish reaction when approaching a barrier and the specification of its swimming, installed a sensor in fish body or installed a camera.

MATERIALS AND METHODS
FLUENT numerical model is the best tool for the numerical simulation of fluid flow in complex geometries. The model is written in c programming language and based on the fundamental principles of computational fluid dynamics such as conservation of mass, momentum and energy and Navier-Stokes equations were used for slow flow analysis and Reynolds stress equations were used in solving the Navier-Stokes equations for the turbulent flow analysis and the finite volume method was used for separation of these equations. This software has the possibility to receive networks with triangular and square elements for two-dimensional geometries and tetragonal, hexagonal and pyramidal networks for three-dimensional geometries (Ismaili et al., 2010). Equations ruling the fluid motion for incompressible turbulent flow including the continuity and momentum equations as relations (1) and (2) are stated that are known as Navier-Stokes equation (Rostami et al., 2008).

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  \hspace{1cm} (1)

\[ \rho \frac{\partial}{\partial x_j} (u_i u_j) = - \frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \]  \hspace{1cm} (2)

That \( u_i \) is the velocity in the \( x_i \) direction, \( \rho \) the general pressure, \( g_i \) is acceleration of gravity in the \( x_i \) direction and \( \tau_{ij} \) is the stress tensor that in the turbulent flow, it can be expressed as the following equation:

\[ \tau_{ij} = \rho (v + v_t) \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \rho (k + \nu_t) \frac{\partial u_i}{\partial x_j} \delta_{ij} \]  \hspace{1cm} (3)

That in the above equation \( v \) and \( v_t \) represents the kinematic viscosity of the fluid viscosity and the turbulent kinetic, \( k \) is turbulence kinetic energy and \( \delta_{ij} \) is coronary Delta.
The side walls are on both sides of the fishway and its elevation is higher than the spillway elevation, so that when a fish jumps out of the spillway will be prevented from getting out of it. The wall thickness is 0.3 meters. Under the bridge and for almost 3 primary ponds, wall thickness is 1 meter. Below the median wall on one side of the weir, an orifice is considered for the passage of fish. This orifice changes direction alternatively. In this case, if in the first weir, orifice is located on the right side, in the second weir, it will be located on the left. The boundary conditions determine the flow and its properties in the solution field, according to the physics of the problem. The applied boundary conditions in the model match the Figure 3. Flow inlet is modeled using velocity inlet boundary condition. At the downstream, we have the outlet boundary of the flow that will be directed to the out of the solution in field without any hindrance. So, the outlet is modeled by boundary condition Pressure outlet. This boundary condition requires defining a partial pressure which by considering the partial pressure equal to zero, the pressure in the outlet will be equal to atmospheric pressure. The free surface of the fluid with the boundary condition (Wall) was modeled. In the context of the fishway, to limit the fluid areas with solid boundary, the wall boundary condition (Wall) was used. Ponds blocks were modeled using the boundary condition (Wall).

![Fig. 3. Boundary conditions for modelling the fishway](image)

**RESULTS AND DISCUSSION**

Software FLUENT uses finite volume method for the discretization of differential equations and converting them into algebraic equations. Software introduces two numerical methods or so-called solver for the solution of algebraic equations. Density based solver is used for high-speed compressible flows and pressure based solver is used for low speed and incompressible flows. Given the discussion above, fluid (water) is an incompressible flow and its movement is done with relatively low speeds, so in this paper, the pressure-based solver was selected for the numerical solution of problems. In this study, the fishway of Karkhe diversion dam - Hamidieh was stimulated using software Fluent. For validation of numerical models, three turbulence models, \( k-\varepsilon \) Standard, \( k-\varepsilon \) RNG and \( k-\varepsilon \) Realizable were used. As can be seen in Figure 4 the results obtained from the turbulence model, \( k-\varepsilon \) Standard is closer to the experimental results. Furthermore, the convergence of solutions of turbulence model, \( k-\varepsilon \) Standard was shorter than the other models. So for the simulation of flow in the fishway of Karkhe diversion dam – Hamidieh, this turbulence model was selected.

![Figure 4: Comparison of velocity in the ponds at different turbulence models with experimental model](image)
As can be seen, the diagram related to the $k-\varepsilon$ Standard and diagram of the experimental model were relatively matched, but the speed range of the numerical model is higher than the rate of experimental models. Flow relative uniform reduction is probably due to the reduction of turbulence near the center of the pond. In Figure 5, flow velocity vectors are seen in the fishway. The blue area which is formed in the middle of any pool represents the ranges of minimum speeds and return and rotating flows that is a good place for fish to relax in each pool.

Figure 5: flow velocity vectors in the fishway
One of the factors that cause energy dissipation in the fishway type pool and weir is the circulating flow in each pool. The velocity vector, the middle part of the pool and near to the middle wall of the upstream is the safe area for fish aggregation and thus it is optimal economically to transfer fish.

CONCLUSION
In this study, first we simulated the flow with three different turbulence models and choosing the optimum model using a simple meshing. Finally, $k-\varepsilon$ Standard model showed more efficiency in flow numerical modelling in the weir and orifice fishway because of modeling the effect of spin than other models and similarity of answers to the laboratory models. The results also indicated that with respect to the velocity vector, the middle part of the pool and near to the middle wall of the upstream is the safe area for fish aggregation and thus it is optimal economically to transfer fish.

REFERENCES
Shamlou H and Aknouni S. 2012. Three-dimensional numerical simulation of flow in the fish pond and overflow using software FLUENT. the first international conference and national conference dam and hydroelectric plants, Tehran.(In Persian)