

## CFD Analysis of Ilisu Dam Sluice Outlet

M. Cihan Aydın<sup>1\*</sup>, Ali Emre Ulu<sup>1</sup>, Çimen Karaduman<sup>1</sup>

<sup>1</sup>Department of Civil Engineering, Bitlis Eren University, Bitlis, TURKEY  
\*mcaaydin@gmail.com

(Received: 18.02.2017; Accepted: 04.03.2017)

### Abstract

Ilisu Dam is one important energy projects in Turkey. After it is completed, it is expected that it will contribute to the country economy approximately \$300 million annually. One of the most remarkable engineering designs of the dam is the conversion of diversion tunnels of 12 m in diameter and about 1 km in length into sluice outlet structure. In this study, the CFD simulation of Ilisu Dam sluice outlet were performed with 1/40 scale. The obtained CFD results were compared with physical model observations conducted by the State Water Works (SWW). A good agreement was achieved between both results, and some useful results of CFD were presented for design of the outlet.

**Keywords:** Ilisu Dam; Sluice Outlet; CFD.

## Ilisu Barajı Dipsavağının HAD Analizi

### Özet

Ilisu Barajı Türkiye'deki en önemli enerji projelerinden biridir. Tamamlandığında ülke ekonomisine yaklaşık olarak yıllık 300 milyon dolar katkı sağlaması beklenmektedir. Barajın en dikkat çekici mühendislik tasarımlarından biri, 12 m çapında ve yaklaşık 1 km uzunluğundaki derivasyon tünellerinin dipsavağa dönüştürülmesidir. Bu çalışmada, Ilisu Baraj dipsavağının CFD simülasyonu 1/40 ölçek ile gerçekleştirilmiştir. Elde edilen CFD sonuçları Devlet Su İşleri (DSİ) tarafından yapılan fiziksel model gözlemleri ile karşılaştırılmıştır. Her iki sonuç arasında iyi bir mutabakat sağlanmış ve CFD'nin bazı kullanışlı sonuçları dipsavak tasarımında sunulmuştur.

**Anahtar Kelimeler:** Ilisu Barajı; Dipsavak; HAD.

### 1. Introduction

Sluice outlet is a hydraulic structure which is designed to completely discharge the dam when necessary, to reduce the spillway capacity and to release the water to be left to the downstream of the river. The sluice outlet can be of different shapes and lengths depending on the needs or shapes of the dams. An important consideration when designing the sluice way is to prevent cavitation damage. In order to avoid this problem engineers may choose different solutions in the sluice outlet design. Experimental studies are conducted to observe how the cavitation phenomenon will actually have an effect. However, these experiments may be time consuming, costly and require a lot of attention. Instead, using the developed computational fluid dynamics (CFD) is now a more functional and reliable method that has been tested many times.

There are many studies exist in the literature that was performed using the CFD. Some of those studies can be seen as follows:

Khan et al. [1] establishes a 3D CFD model of The Dalles Dam forebay. The aim of the model is to investigate the effects of clogging resulting from the accumulation of debris in the 12.3 m section above the turbine intake. The model includes approximately 0.80 million cells consisted from 22 power plant units, two fish turbine units, station services unit and a forebay bathymetry. The CFD model has quite well simulated the velocity distributions observed in a physical model. Numerical simulations indicated that blocked garbage pits would change velocity distributions around the powerhouse to a significant extent [1]. Cassan and Belaud [2] defines the flow properties at a large opening to improve the discharge calculation for submerged channel gates. To do so, the technical note

investigated the upstream and downstream flow characteristics of the channel inlet gates experimentally and numerically by means of a fluid method using average Navier-Stokes two-dimensional simulations by Reynolds [2]. Zhang et al. [3] develops a dynamic numerical model to investigate a laboratory experiment. Through analysis and modeling of the observed data, the river water quality and quantity (WQQ) operations under the water channel regulation were examined and the interaction between the WQQ and the regulatory capacity downstream of a water channel was investigated [3]. Li et al. [4] developed a numerical model to simulate the flow of probable maximum flow (PMF) on a system consisting of an existing service spillway and a new auxiliary spillway. In the study, approach channel geometries consisted from different combinations were simulated. The article demonstrates the successful implementation of a CFD model in the design process of an auxiliary spillway and encourages the hydraulic engineers and CFD modelers for designing the hydraulic structures [4]. Ebner [5] investigated CFD model used applications to reveal its limitations and to make some assumptions. At the end, article noted that CFD numerical models are a good way to recognize the hydraulic conditions [5].

A technical report modified a previously computed fluid dynamics model, and it was used to characterize tailrace hydraulic and sluice outlet exit conditions for low total river and low level spillway flows [6]. Kökpınar and Çelik [7] investigated the preliminary test results obtained from a large scale Deriner Dam of tunnel spillway model are presented. A number of test models have also been carried out for the

original and final project cases. In the article it was seen that the original designed ventilator did not work effectively from the values obtained according to the original project status. For this reason, it became necessary to change the original progeny in terms of location and geometry of the aerator [7].

From the current literature, it can be said that CFD models are valuable, useful and reliable tools for the hydraulic structures. As one can understand that there is limited study about the CFD models of the sluice outlet. This study investigates the working principle of sluice outlet of Ilisu Dam by using CFD model.

## 2. Ilisu Dam and HPP

Ilisu Dam and Hydroelectric Power Plant (HPP) is one of most important energy project with 1200 MW installed capacity in Turkey. After completed, the total produced energy will be 3833 GWh/year, and it is expected to bring about \$300 million incomes to the economy annually. This energy corresponds to 10% of the hydroelectric energy to be produced in Turkey. The general project characteristics of Ilisu Dam and its sluice outlet structures were given in Table 1 [8], [9].

DT2 derivation tunnel of Ilisu Dam was converted to the outlet structure. DT1 derivation tunnel parallel to DT2 tunnel also were designed as an aeration tunnel for air supply purpose. Two tunnels were connected by a horseshoe cross-section air gallery. The detail of outlet structures converted from DT2 derivation tunnel were showed in the Fig. 2.

**Table 1.** Ilisu Dam and HPP Project Characteristics

Type of Dam	Embankment, concrete-face rock-fill
Purpose	HP, flood control and irrigation
Status	Under construction
Location	Dargeçit County, between Mardin and Şırnak
River	Dicle River
Construction cost	1.7 billion \$
Height from thalweg	135 m
Length	1.820 m
Dam volume	43,800,000 m <sup>3</sup>
Installed capacity	1,200 MW
Average annual energy production	6 x 200 MW Francis-type
Hydraulic head	122.6 m
Annual generation	3,833 GWh
Derivation slope	0.1%

Derivation structures	DT1+DT2+DT3 derivation tunnels
Sluice outlet	DT2 derivation tunnel
Diameter of sluice outlet	12 m
Length of the outlet	1016.1 m
Type of control valve	Sluice valve
Number of control valve	2
Dimensions of control valve	2.65x4.00 m
Spillway type	Service overflow, controlled-chute
Spillway capacity	18,000 m <sup>3</sup> /s

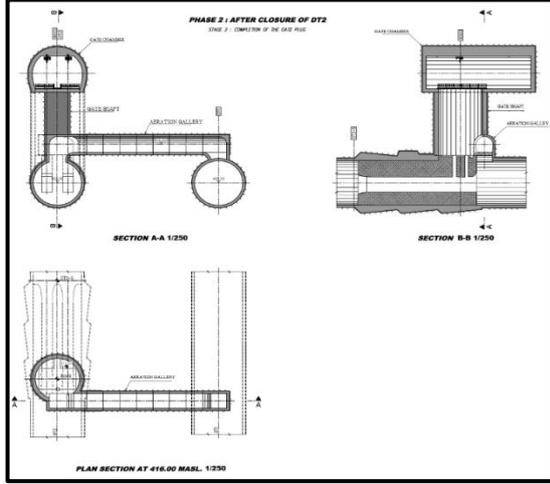


Fig. 1. Details of the sluice outlet converted from DT2 derivation tunnel of Ilisu Dam [8].

### 3. Method

In this study, a three-dimensional (3D) numerical simulation using FLOW-3D software was applied to Ilisu Dam sluice outlet to

$$\begin{aligned}
\frac{\partial u}{\partial t} + \frac{1}{V_F} \left[ uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right] &= -\frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x - \frac{R_S}{\rho V_F} (u - u_w - \delta u_s) \\
\frac{\partial v}{\partial t} + \frac{1}{V_F} \left[ uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right] &= -\frac{1}{\rho} \frac{\partial p}{\partial y} + G_y + f_y - \frac{R_S}{\rho V_F} (v - v_w - \delta v_s) \\
\frac{\partial w}{\partial t} + \frac{1}{V_F} \left[ uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right] &= -\frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z - \frac{R_S}{\rho V_F} (w - w_w - \delta w_s)
\end{aligned} \quad (2)$$

where;  $G_x$ ,  $G_y$ ,  $G_z$  are body acceleration components,  $f_x$ ,  $f_y$ ,  $f_z$  are viscous accelerations components,  $u_w, v_w, w_w$  are the components of velocity source;  $u_s, v_s, w_s$  are the velocity components at the surface.

#### Physical Model

The experimental studies on the revised design of the sluice outlet structure and aeration tunnel were conducted by State Hydraulic Works in Turkey. In the first design of the Ilisu Dam sluice outlet, the transport tunnels to the sluice outlet vane chamber were planned as ventilation galley providing air flow. However, in the tests and

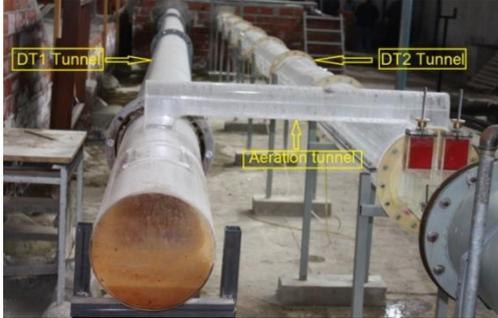
determine the two-phase (air-water) flow properties. FLOW-3D is a general purpose CFD program which is especially effective free surface flows for one and two phase flows. The program solves the mass continuity and the Navier-Stokes equations as the momentum equation for each element to estimate properties of fluid motions. These equations can be given as follows respectively for Cartesian coordinate system [10].

$$V_F \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} (\rho u A_x) + \frac{\partial}{\partial y} (\rho v A_y) + \frac{\partial}{\partial z} (\rho w A_z) = R_D + R_S \quad (1)$$

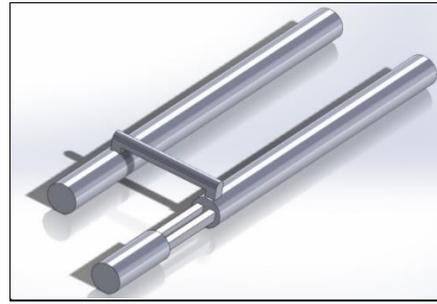
where,  $V_F$  is volume fraction,  $\rho$  is the density of fluid;  $R_D$  is a turbulent diffusion term;  $R_S$  is a mass source;  $A_x$ ,  $A_y$ , and  $A_z$  are the fractional areas in the  $x$ ,  $y$  and  $z$ ;  $u$ ,  $v$  and  $w$  are velocity components. The Navier-Stokes are used to describe three-dimensional fluid dynamics.

analyses conducted, it was understood that the air in the narrowing sections would reach a speed as high as 100 m/s and eventually air explosions and operational problems would occur. For this reason, one of the two sluice outlet (DT1) was planned as an air gallery with a diameter of 12 m and a length of about 1000 m, and the air for the DT2 sluice outlet was provided by an air gallery with a horseshoe section here. Different alternatives of this system have been experimentally investigated by DSI with 1/40 model and solution proposal has been introduced. The experimental setup of the outlet

and aeration tunnels performed by DSI was showed in Fig. 2.



**Fig. 2.** Experimental setup of the outlet and aerations tunnels [8]



**Fig. 3.** Geometry of 3D numerical model

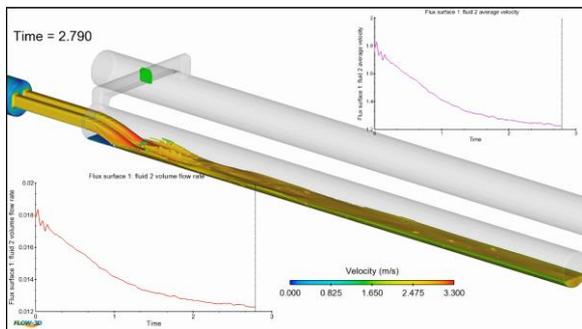
*Numerical Model*

The 3D numerical model was given in the Fig. 3. Numerical model was divided into 3,900,000 structured 3D hexahedral elements. The simulation duration is approximately 7 hours with two real core xenon processor and 8 GB ram. Two phases model with *renormalized group (RNG)* turbulence model were used in the numerical solution. Numerical model with 12 m diameter of DT1 and DT2 tunnels were scaled by 1/40 scale similar to the experimental model. The dimension of the rectangular sluice section is 2.65x4.00m. The height and width of the horseshoe air tunnel are 4.5 m and 4.20 m respectively.

The CFD analysis were performed for 24.75 m of the hydraulic head and 430 m<sup>3</sup>/s of water outlet discharge with laboratory model dimensions (1/40 scale). The solution convergence was achieved in 5.0 second. The solution convergence curves and water velocity of the sluice flow were illustrated in Fig. 4. As seen at the convergence curves in Fig. 5, the air flow rate in the aeration tunnel is almost steady while approaching to 0.012 m<sup>3</sup>/s and velocity of 1.2 m/s. This corresponds 121.43 m<sup>3</sup>/s and 7.58 m/s of prototype values. The air entrainment discharges and average air velocity were observed by DSI as 118.76 m<sup>3</sup>/s and 7.35 m/s experimentally [8]. The air entrainment rates ( $\beta=Q_a/Q_w$ ) are calculated as 0.303 of experiment, and 0.309 of CFD. The relative percent error between experimental and numerical (CFD) results is approximately 2%. These comparisons were given in Table 2.

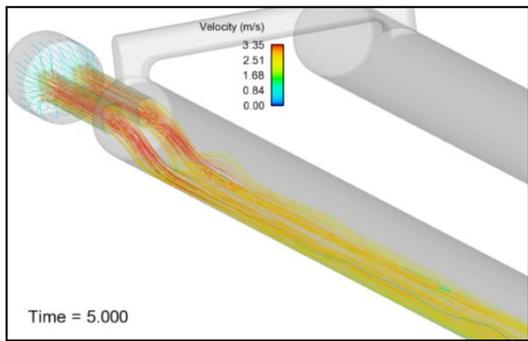
**Table 2.** Comparison of experimental and CFD results

	Hydraulic Head, $H_t$ (m)	Water discharge, $Q_w$ (m <sup>3</sup> /s)	Air entrainment discharge, $Q_a$ (m <sup>3</sup> /s)	Average Air Velocity, $V_a$	Air entrainment coefficient ( $\beta=Q_a/Q_w$ )
Experiments	24.75	392	118.76	7.35	0.303
CFD	24.75	392	121.43	7.58	0.309
Error (%)			2.2	3.0	2.0



**Fig. 4.** Solution of the numerical model with convergence curves.

The maximum velocity of water flow was observed in jet near to impact region at the downstream of sluice gate. The streamlines of the water flow are shown in the Fig. 5. This figure also indicates same the maximum water velocity position.

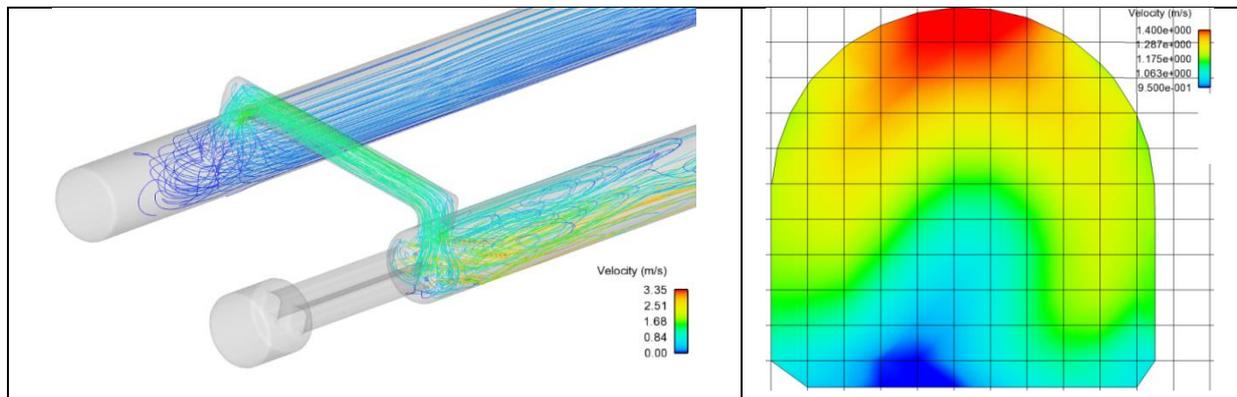


**Fig. 5.** Streamlines of the water flow in the sluice outlet

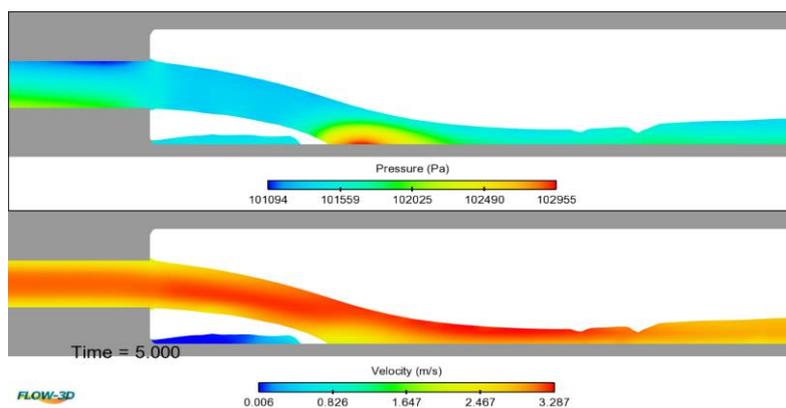
Fig. 6 shows the streamlines of air flow (on the left) and velocity contours at the middle section of aeration tunnel (on the right). It is observed in these figures that the air velocity on the section is irregular and the maximum

velocity occurs at the top of the section while the velocity at a lower part of the section was almost zero. The maximum air velocities in the DT2 tunnel were observed near to water surface probably owing to slip velocities between air and water flow.

In Fig. 7, the pressure and velocity contours on the water flow profiles are given with colored scales. In the upper figure, the maximum pressure occurs at the impact point of the water jet while the minimum pressure is formed at the upper of the sluice. The velocities inside (in the lower figure) the sluice is higher than that of the edges of the sluice due to the wall effects.



**Fig. 6.** Streamlines of air flow (on the left), and the velocity distribution at the cross-section of the aeration tunnel (on the right).



**Fig. 7.** Pressure and velocity contours of water flow jet downstream of sluice gate.

#### 4. Conclusions

In this study, Iısu Dam's sluice outlet, which have 12 m diameter, was analyzed by using CFD

simulation technic, and was discussed. The results of CFD analysis were compared to the experimental tests with 1/40 scale performed by DSI, and a good agreement was achieved in

terms of air entrainment by air supply tunnel. The CFD results are reasonably compatible with the experimental observations with the percent errors from 2.00 to 3.00%. The experimental study is more expensive and required more effort than CFD analyses. But still, the CFD models need to be calibrated with some experimental data. The calibrated numerical models may be more useful and easier to determine hydraulic characteristics than experimental studies. The results encouraged the further researches for detailing the design of the Ilisu Dam outlet and the other similar projects.

## 5. Acknowledgement

- This study has been supported by Scientific Research Project Department of Bitlis Eren University in Turkey with Project No: BEBAP 2017.04.
- We thanks to Firat University Department of Civil Engineering, the State Water Works (SWW) and IOG Engineering for their valuable contribution.
- This paper is also available in Oral Submission Book of International Conference on Advances and Innovations in Engineering (ICAIE) May 10-12 2017, Elazığ.

## 6. References

1.Khan, L.A., Wicklein, E.A., Rashid, M., Ebner, L.L. and Richards, N.A. (2008). Case study of an application of a computational fluid dynamics model

to the forebay of the Dalles Dam, Oregon, *Journal of Hydraulic Engineering*, **134(5)**: 509-519.

2.Cassan, L. and Belaud, G. (2011). Experimental and numerical investigation of flow under sluice gates. *Journal of Hydraulic Engineering*, **138(4)**, 367-373.

3.Zhang, Y., Xia, J, Shao, Q and Zhang, X. (2011). Experimental and simulation studies on the impact of sluice regulation on water quantity and quality processes. *Journal of Hydrologic Engineering*, **17(4)**: 467-477.

4.Li, S., Cain, S. Wosnik, M., Miller, C., Kocahan, H and Wyckoff, R. (2010). Numerical modeling of probable maximum flood flowing through a system of spillways, *Journal of Hydraulic Engineering*, **137(1)**: 66-74.

5.Ebner, L.L: (2014). Practical Use of Computational Fluid Dynamic Models as a Design Tool-Limitations and Assumptions. *Critical Transitions in Water and Environmental Resources Management*, 2004, pp. 1-10.

6.Rakowski, C.L, Richmond, M.C. Serkowski, J.A. and Perkins, W.A. (2010). Computational Fluid Dynamics Modeling of the Bonneville Project: Tailrace Spill Patterns for Low Flows and Corner Collector Smolt Egress (No. PNNL-20056), Pacific Northwest National Laboratory, (PNNL), Richland, WA (US).

7.Kökpınar, M.A. and Çelik, H.Ç. (2002). Deriner Barajı Tüneli Dolusavak Havalandırıcıları Büyük Ölçekli Hidrolik Model Çalışmaları.

8.DSİ, (2013). Ilisu Barajı ve HES Projesi Dipsavak Tüneli Ek Hidrolik Çalışmaları (M-398), Fiziksel Model Deney Raporu, DSİ TAKK Dairesi Başkanlığı Hidrolik Model Laboratuvarı Şube Müdürlüğü. Ankara. Rapor No: Hİ-1022, 119 s.

9.Wikipedia, (2017), Ilisu Dam. [https://en.wikipedia.org/wiki/Ilisu\\_Dam](https://en.wikipedia.org/wiki/Ilisu_Dam), Accessed date: 26 Feb. 2017.

10.FLOW-3D (2016). User Manual, Theory Guide