www.dergipark.gov.tr
ISSN:2148-3736
El-Cezerî Fen ve Mühendislik Dergisi
Cilt: 5, No: 3, 2018 (856-861)

El-Cezerî Journal of Science and Engineering
Vol: 5, No: 3, 2018 (856-861)
DOI : 10.31202/ecjse.432646

ECJSE

Research Paper / Makale

Numerical Investigation of Nappe Flow Regime in Stepped Spillway

Tamer BAĞATUR[a], Ömer Faruk CAN[b]

[a]Dicle University, Engineering Faculty, Department of Civil Engineering, 21280, Diyarbakir, TURKEY
[b]Dicle University, Engineering Faculty, Department of Mechanical Engineering, 21280, Diyarbakir, TURKEY

Received/Geliş: 10.06.2018 **Revised/Düzeltme:** 23.07.2017 **Accepted/Kabul:** 28.07.2017

Abstract: The stepped spillways consist of concrete steps on the spillway discharge channel to assist in the dissipation of the kinetic energy of the water flow. This structure reduces size or eliminates the need for an additional energy dissipator on downstream toe. In this study, an investigation of nappe flow regime has been carried out to study the influence of operating conditions on the hydrodynamics and mass transfer characteristics on the steps of the spillway by using two-fluid computational multiphase fluid dynamics (CMFD) modeling. The numerical work was performed by means of the program ANSYS CFX 11.0 software. The CMFD results are validated with experimental observation. Thus, the numerical performance of nappe flow is analyzed.

Key words : Stepped spillway, Nappe flow regime, Air-water, Computational Fluid Dynamics (CFD).

Basamaklı Dolusavaklarda Nap Akımının Nümerik İncelenmesi

Öz : Basamaklı dolusavaklar, baraj yapılarının dolusavak ünitesinden salınan suyun kinetik enerjisini sönümlemeye yardımcı olmak için dolusavak şutu üzerinde yapılandırılan basamaklardan oluşur. Bu yapı, dolusavak boyutunu azaltır veya dolusavak sonunda gerek duyulan ek bir enerji kırıcı ihtiyacını ortadan kaldırır. Bu çalışmada, iki akışlı hesaplamalı çok fazlı akışkanlar dinamiği (CMFD) modellemesi kullanılarak, çalışma koşullarının hidrodinamik ve kütle aktarım özellikleri üzerinde akışkan etkisini incelemek amacıyla nap akış rejimi için bir sayısal araştırma yapılmıştır. Sayısal çalışma, ANSYS CFX 11.0 bilgisayar paket programı ile gerçekleştirilmiştir. Sayısal sonuçlar deneysel gözlemle doğrulanmıştır. Sonuç olarak, nap akımının sayısal performansı analiz edilmiştir.

Anahtar kelimeler: Basamaklı dolusavak, Nap akım rejimi, Hava-su, Hesaplamalı akışkanlar dinamiği.

1. Introduction

In nowadays due to prefering roller compacted concrete (RCC) type dams and prefabricated concrete blocs, stepped spillways have been more popular for design of dams in the worlwide. Stepped spillway structures reduce the need of large energy dissipater basins at the toe of the spillway. The flow over a stepped spillway or channel may be divided into three flow types called nappe, skimming and transition depending on the flow discharge for a given stepped spillway structure. Nappe flow regime dissipates maximum energy and oxygen transfer for small size stepped spillway. The energy dissipation on stepped spillway depends upon step number, height, length and unit discharge. In literature, the studies and information related to stepped spillway and channel are available such as [1], [2] and [3]. Figure 1 shows nappe flow regime in stepped channel.

How to cite this article
Bağatur, T., Can Ö. F., "Numerical Investigation of Nappe Flow Regime in Stepped Spillway", El-Cezerî Journal of Science and Engineering, 2018, 5(3); 856-861.

Bu makaleye atıf yapmak için
Bağatur, T., Can Ö. F., "Basamaklı Dolusavaklarda Nap Akımının Nümerik İncelenmesi", El-Cezerî Fen ve Mühendislik Dergisi 2018, 5(3); 856-861.

Figure 1. Stepped channel in nappe flow condition

Computational fluid dynamic (CFD) method or tecnique is widely used in research due to an extensive range of features in different fields of science and engineering to review the performance of system or design [4]. ANSYS CFX is one of the foremost tools of CFD, which uses the finite volume technique to study the complex phenomenon such as multiphase flow, chemical reactions, turbulence and heat and mass transfer in system and design. This method provides the capability to use different physical models such as incompressible or compressible, inviscid or viscous, laminar or turbulent, etc. The k–ω model model can be used in the simulations [5] (see ANSYS CFX Solver Theory Guide, 2011).

Chanel et al. studied on assessment of spillway structures modeling using computational fluid Dynamics approach [6]. Aydin and Ozturk developed a computational fluid dynamics model for mechanism of air entrainment at spillway aerators [7]. Chinnarasri et al. simulated model for flow over spillways using computational fluid Dynamics [8]. Also, Dursun and Ozturk investigated flow characteristics of stepped spillways using numeric model [9].

This paper introduces a numerical and experimental study of nappe flow regime, the hydrodynamics and mass transfer characteristics on the steps of the spillway by using two-fluid computational multiphase fluid dynamics modeling. The numerical work is performed by means of the ANSYS CFX 11.0 (Academic Teaching Introductory version) computer
program. The CMFD results are validated with experimental data. Thus, numerical performance of the steps for nappe flow regime is analyzed.

2. Experimental Description and Data

The data used in this study were selected and compiled from studies conducted by [10] on a large model of a stepped channel. All experiments efforts were conducted in a rectangular channel 0.30 m width and 0.50 m height. All experimental studies were completed in unit discharges (q) ranging between 16.67 and 166.67 litres/s/m. For the stepped channel studies, the downstream channel was 3.0 m long, 0.35 m width and 0.45 m height. Step height of channel (h) is equal to 0.10 m. The slope of the stepped channel was selected as 30^o.

In experimental studies, nappe flow regime is observed for 16.67 litres/s/m unit discharge. Thus, numeric analysis is reviewed for this unit discharge in this study.

3. General Methodology for Developing a CFD Model

There are four basic approaches involved in developing a CFD model as shown in Figure 2. The first step is the design of the model geometry. ANSYS have its own geometry builder. Otherwise, the structure geometry of system can be drawn with any computer aided design software and then imported into the ANSYS module. The second step is creating the mesh. Meshing is the process of describing the structure using mesh of different shapes and sizes, as cubes, prism, tetrahedral, hexacore, or hybrid volumes. For each of these mesh units, the hydraulic particulars are computed using the numerical method in the ANSYS. The developed mesh is then imported into the ANSYS and the solution is obtained using different solvers depending on the type of analysis. Finally, the results are analyzed using a separate graphical analysis tool in ANSYS. Alternatively, the results can also be analyzed within ANSYS which offers some limited functionalities.

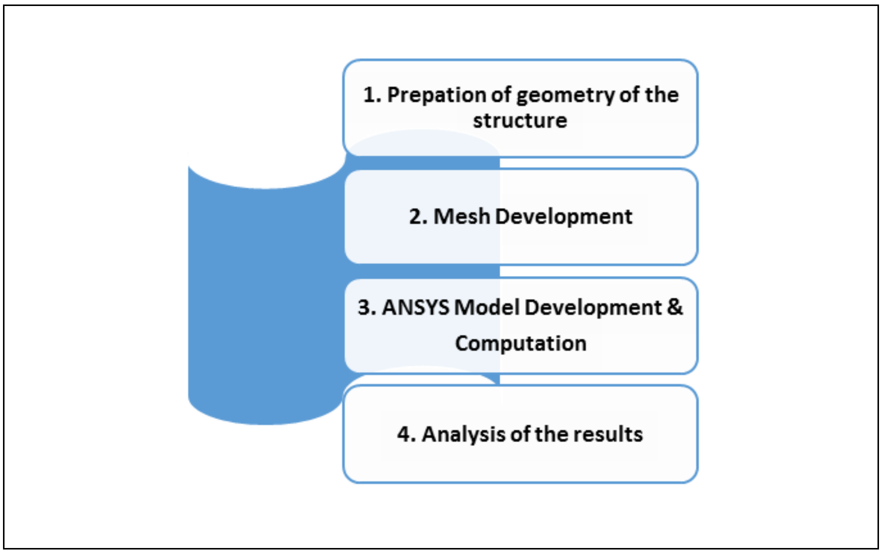

Figure 2. General methodology for numeric model of ANSYS software

4. Boundary Conditions of Stepped Channel

The boundary limits for the numerical model of nappe flow study are examined as flow velocity (V), pressure (P) and walls (WALL-solid surfaces). The velocity inlet boundary condition is applied to determine the water flow rate at the entry of stepped spillway. Boundary conditions of the numerical study for stepped spillway and channel are given in fig.3.

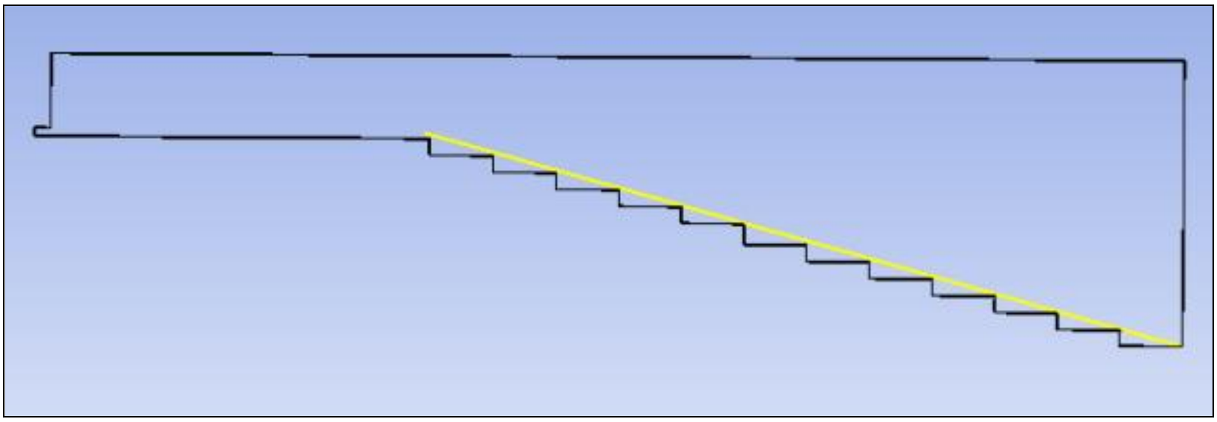

Figure 3. The model of stepped spillway and its channel according to boundary conditions

To obtain the required flow field, solutions have been obtained by solving the basic governing equations with inlet velocity normal to main inlet and side inlet as boundary conditions or input parameters. Further, no slip condition was imposed at the solid surfaces. The characteristics of flow was considered to rep-resent water by imposing a density and viscosity value of 997 kg/m^3 and 0.001 Pa s.

4.1. Grid and Iterative Convergences for numeric model

Geometry and grid generation is carried out using ANSYS workbench which is the preprocessor bundled with ANSYS CFX. In this paper, the numerical model geometries are prepared with ANSYS workbench program and are divided into 34712 node numbers and 27809 element numbers (Fig. 4). The fine and coarse grids are processed in the flow domain and the convergence and stability of the solution are onteined to be insensitive to the grid size in the main body of the nappe flow for stepped channel.

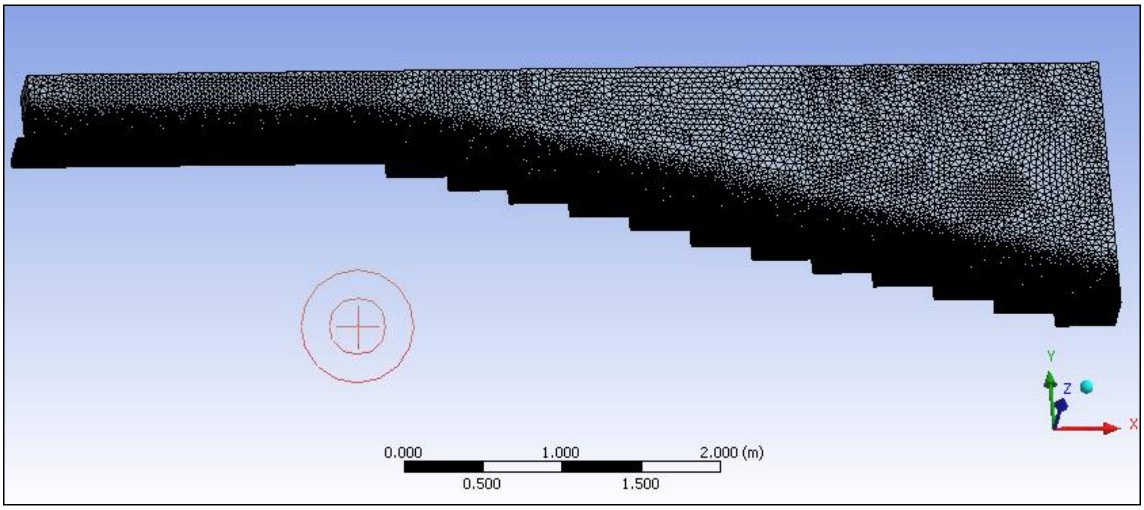

Figure 4. Meshing model for computational domain of stepped spillway

In the iterative solution cases, it must be ensured that iterative convergence is done with at least three orders of magnitude decrease in the normalized residuals for each equation solved. Convergence values were taken as 1x10-6. Iteration number was taken as 1000. However, values were converged to the value 1x10-6 approximately at 150-200 iterations.

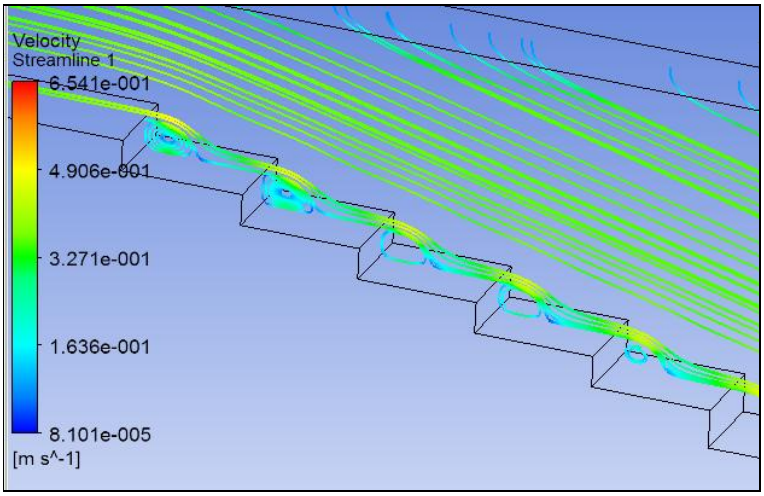

Figure 5. Development of nappe flow regime on stepped channel (q=16.67 litres/s/m)

5. Results of Numeric Model Study for Stepped Channel

Using computational multiphase fluid dynamics modeling, nappe flow type is analyzed. This analysis is done by means of the computer package program ANSYS CFX SOFTWARE that processes finite volumes theory. Fig. 5 presents development of nappe flow regime on step. Fig.6 shows velocity vactors on a step of spillway discharge channel. The turbulence and mixture movements develop on the step and corner. In experimental studies, nappe flow is realized for unit discharge equal to q=16.67 litres/s/m. As seen in the figures, nappe flow is numerically presented for same unit discharge. Also, fig. 7 shows pressure contours on stepped channel.

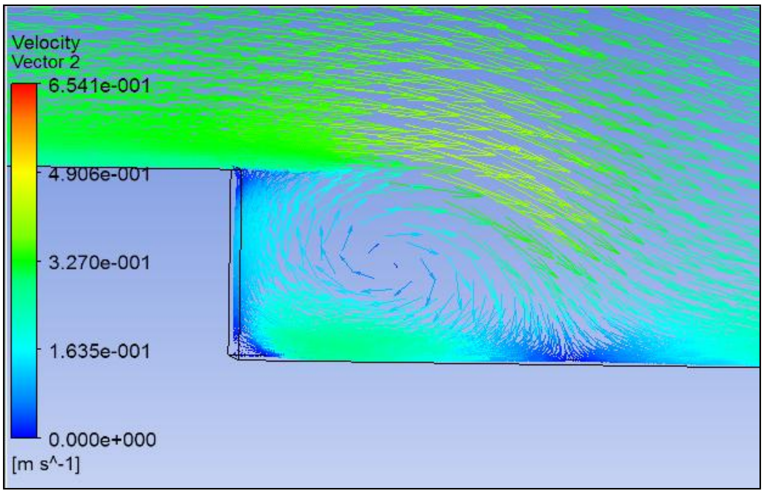

Figure 6. Velocity vactors of flow on a step (q=16.67 litres/s/m)

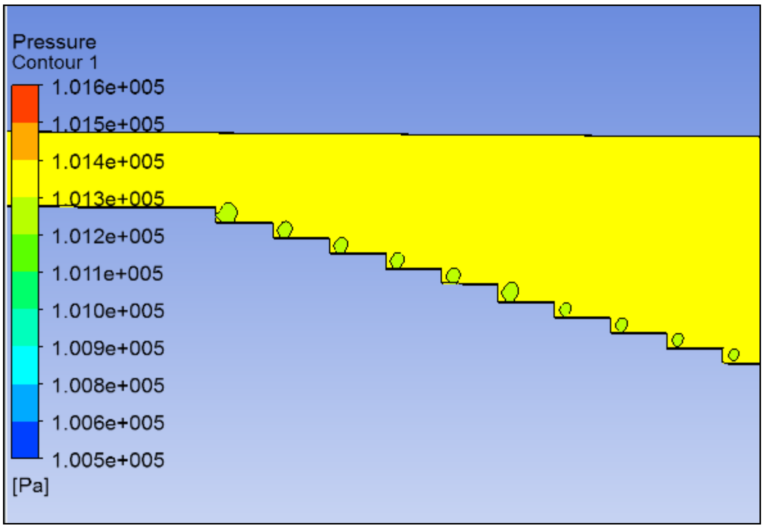

Figure 7. Pressure contour on stepped channel (q=16.67 litres/s/m)

2. Conclusions

The computational multiphase fluid dynamics (CMFD) technique has low cost, and its computation period is shorter than experimental period and studies, so the system parameters can be computed as much as possible. At the same time, the numerical simulation involves the concept of field, which is different from the rough mode of traditional design. It can not only reflect the practical flow characteristics on stepped spillwy and its discharge channel but also obtain the relation between the self-sucking flowrate and structural parameters and therefore provide reference for the parameters design. Numerical simulation can predict the distribution of the pressure and velocity comparatively precisely, and help to further improve nappe flow of stepped spillway. In present paper, nappe flow regime of stepped spillway were analyzed using CMFD modeling. Numerical analysis was carried out by means of the ANSYS CFX program. In experimental studies, nappe flow regime is observed for unit discharge equal to q=16.67 litres/s/m. As seen in the numeric model studies, nappe flow type is numerically presented for same unit discharge value. Thus, this study presents prediction of nappe flow conditions for stepped spillway.

References

[1]. Chanson H., "The Hydraulics of Stepped Chutes and Spillways", Balkema, Lisse, The Netherlands, 2002.

[2]. Chen, Q., Dai, G. and Liu, H., "Volume of Fluid Model for Turbulence numerical simulation of stepped spillway overflow", *Journal of Hydraulic Engineering*, 2002, 128 (7): 683–688.

[3]. Boes, R.M. and Hager, W.H., "Hydraulic Design of Stepped Spillways", *Journal of Hydraulic Engineering*, 2003,129 (9): 671-679.

[4]. Ferziger, J. H., Peric, M., "Computational Methods for Fluid Dynamics", *Springer-Verlag*, Berlin, Heidelberg, 1996.

[5]. ANSYS CFX Solver Theory Guide, 2011.

[6]. Chanel, P. G., and Doering, J. C., "Assessment of Spillway Modeling Using Computational Fluid Dynamics", *Canadian Journal of Civil Engineering*, 2008, 35 (12): 1481–1485.

[7]. Aydin, M. C., and Ozturk, M., "Verification and Validation of a Computational Fluid Dynamics (CFD) Model for Air Entrainment at Spillway Aerators", *Canadian Journal of Civil Engineering*, 2009, 36 (5): 826–836.

[8]. Chinnarasri, C., Kositgittiwong, D., and Julien, P. Y., "Model of Flow Over Spillways by Computational Fluid Dynamics", *Proceedings of the Institution of Civil Engineers - WaterManagement*, 2014, 167(3): 164–175.

[9]. Dursun, O. F. and Ozturk, M., "Determination of Flow Characteristics of Stepped Spillways." *Proceedings of the Institution of Civil Engineers - Water Management*, 2016, 9 (1): 3–42.

[10]. Baylar A., Emiroglu M. E. and Bagatur T., "An Experimental Investigation of Aeration Performance in Stepped Spillways". *Water and Environment Journal*, 2006, 20 (1): 35–42.