Numerical Modeling of Flow in Side Channel Spillway Using ANSYS-CFX

Shahin S. Ahmed¹ and Yaseen W. Aziz²

¹ Asst. prof. Department of Dams and Water Resources, College of Engineering, University of Salahaddin, Erbil, Kurdistan Region, Iraq.
² M.Sc. Department of Dams and Water Resources, College of Engineering, University of Salahaddin, Erbil, Kurdistan Region, Iraq.

ARTICLE INFO

Article History:
Received: 14/05/2017
Accepted: 12/04/2018
Published: 01/06/2018

Keywords:
Side Channel Spillway
CFD
ANSYS-CFX
Turbulence models

*Corresponding Author:
Email: yaseen90aziz@gmail.com

ABSTRACT

Side channel spillway is one of the most common types of spillways provided at earth dams to release flood discharge laterally. Flow in side channel is complex due to discharge changing along its length. Due to rapid advent of computer technology computational fluid dynamics (CFD) is extensively used to model and analyze complex issues in engineering problems. In the present study ANSYS-CFX code has been used to predict flow characteristics in the side channel spillway. Grid dependence study provided to select the optimum grid size that can predict free surface profile accurately with minimum computational time. In addition, various turbulence models such as k-ε and RNG k-ε were used. The capability of each turbulence model to predict flow characteristics in the spillway were tested. The k-ε and RNG k-ε turbulence models gives good results at most parts of the spillway, while RNG k-ε turbulence model needs higher computational time. Furthermore, the ability of the code to predict free surface water profile for various discharges was validated using results of the physical model. The comparisons of water surface profile predicted by ANSYS-CFX with the physical model show good agreement. Unlikely, at the location of hydraulic jump in the stilling basin for higher discharges characterized by strong aeration ANSYS-CFX with k-ε turbulence model failed to give accurate predicts of the free surface water profile.

INTRODUCTION

Side channel Spillway (SCS) is one of the types of spillway that is usually provided at earth dams in order to release flood discharge from the reservoir to prevent overtopping and damage of the dam. Therefore, spillway is an important structure in the dam.
mechanics that can simulate real fluid by using numerical methods to solve the governing equations. Nowadays CFD is extensively used by the researchers to model and analyze complex problems in engineering due to rapid advent of high performance computers and parallel computation methods. Investigation of dynamic pressure fluctuation in stepped three sided spillway (U shape) using FLOW 3D code was performed by (Taghizadeh et al, 2012). A good agreement was found when the results of the numerical model compared with the experimental results. Furthermore, (Gellibert et al, 2016) conducted a study to investigate the performance of side channel spillway of a selected dam using ANSYS-CFX code. The result of water surface profile in the spillway compared with results of the original study, the results showed good agreement between them. Flow characteristics in the hydraulic jump investigated using ANSYS-CFX code, Flow-3D and openFOAM by (Castillo et al, 2014), the k-ω turbulence model in the recirculation zone displayed a better agreement and more realistic than k- ε. The flow simulation in front of rectangular broad crested weir was carried out by (Zachoval and Rousar, 2015) and compared their results with experimental results. The most reliable result was obtained by using RNG k- ε turbulence model but from all two layers turbulence models (SST) provide most reliable results. In the present study the flow characteristics in SCS were investigated using ANSYS-CFX 14. The results of the numerical model were validated against the physical model data of (Aziz, 2016).

MATERIALS AND METHODS

In the current study ANSYS-CFX code has been used for flow simulation in the side channel spillway. The code is based on finite volume method, which discretizes Naiver stokes equations at each computational cell. In turbulent flow, velocity at each point consists of two components, mean (\( \bar{U} \)) and fluctuating velocities (\( u' \)). The mass and momentum equations in the time average form for incompressible flow can be written as:

\[
\frac{\partial}{\partial x_i} (\rho \bar{U}_i) = 0 \quad \text{............................} \quad 1
\]

\[
\frac{\partial}{\partial x_j} \rho \bar{U}_i \bar{U}_j = -\rho g_i - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \right) - \rho u'_i u'_j \quad \text{...2}
\]

The Naiver - Stokes equations with time average velocity called Reynolds averaged Naiver – Stokes (RANS) equations. This method eliminates turbulent fluctuations by the averaging process. The averaging of nonlinear terms in the Naiver Stokes equations causes additional unknowns called Reynolds stress. Most of commercial CFD codes use time average equations such as RANS equations for modeling turbulent flow. The term (\( -\rho u'_i u'_j \)) is referred to the Reynolds stresses, in three dimensional (RANS) equations there are six unknown terms, they behave like stresses.

The turbulence modeling is a computational procedure that can close the governing equations by modeling Reynolds stresses (Piradeepan, 2002). Numerous turbulence models are available based on RANS equations. ANSYS-CFX contains numerous turbulence models which can be divided into two groups namely; eddy viscosity and Reynolds stress models.

The k – ε Turbulence Model

This turbulence model is a semi empirical model. It is probably the most common type used than the other types. It gives a good result in many industrial flows. This model has two model equations one for k (turbulent kinetic energy) and other for ε (dissipation rate). In this model the eddy viscosity is linked to the
turbulent kinetic energy and dissipation (ANSYS, 2011a) as:

\[
\mu_\varepsilon = \rho C_\mu \frac{k^2}{\varepsilon} \quad \text{.................................. 3}
\]

Where: \( k \) is the turbulent kinetic energy, \( \varepsilon \) is the rate of dissipation of turbulent kinetic energy, \( C_\mu \) is an empirical constant = 0.09.

The separate transport equations solve for \( k \) and \( \varepsilon \) at a given time. The transport equations for \( k \) and \( \varepsilon \) are as follow (Lauder and Spalding, 1974):

\[
\begin{align*}
\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho \text{grad} k) &= \text{div} \left[ \frac{\mu_k}{\sigma_k} \text{grad} k \right] + 2\mu_\varepsilon S_{ij} - \rho \varepsilon \quad \text{.................................. 4} \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \text{div}(\rho \text{grad} \varepsilon) &= \text{div} \left[ \frac{\mu_k}{\sigma_k} \text{grad} \varepsilon \right] + \frac{C_1\varepsilon}{k} - \frac{C_2\rho \varepsilon^2}{k} \quad \text{.................................. 5}
\end{align*}
\]

Where:

\( \sigma_k \): is Prandtl number connect the diffusivity of \( k \) to the eddy viscosity, typically the value of 1.0 is used, \( \sigma_\varepsilon \): is Prandtl number connect the diffusivity of \( \varepsilon \) to the eddy viscosity, typically the value of 1.3 is used. The value of \( C_1\varepsilon \) and \( C_2\varepsilon \) are 1.44 and 1.92 respectively.

The standard \((k-\varepsilon)\) model equations (4), (5) have been developed for fully turbulent flow which cannot be applied to the near wall that characterized by viscous layer has low Reynolds number, this leads to erroneous results (Abo, 2013).

**The Renormalized Group (RNG) k-\( \varepsilon \) Turbulence Model**

The RNG k-\( \varepsilon \) turbulence model is based on the re-normalization group (RNG) analysis of Naiver – Stokes equations (ANSYS 2011a). The RNG method uses statistical mechanics to extend the k-\( \varepsilon \) model. This model systematically removes the small scales of motion from the Naiver - Stokes equations by expressing their effects in terms of large scale motions and modified viscosity (Versteeg and Malalasekera, 2007, p.87).

The RNG k-\( \varepsilon \) model was first derived by (Yakhot et al., 1992), the equations are as follows:

\[
\begin{align*}
\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho \text{grad} k) &= \text{div}(\alpha_k \mu_{eff} \text{grad} k) + \tau_{ij} - S_{ij} \quad \text{.................................. 6} \\
\frac{\partial (\rho \varepsilon)}{\partial t} + \text{div}(\rho \text{grad} \varepsilon) &= \text{div}(\alpha_k \mu_{eff} \text{grad} \varepsilon) + C_1\varepsilon - S_{ij} \quad \text{.................................. 7}
\end{align*}
\]

Where:

\( \alpha_k = \alpha_k = 1.39 \), \( C_{1\varepsilon} = 1.42 \), \( C_2\varepsilon=1.68 \), \( \beta = 0.012 \), \( \eta_o = 4.377 \)

**Geometry and Mesh Generation**

For the present study, three dimensional geometry of the model was created using AutoCAD 2013. The geometry exported from AutoCAD as a file (.sat) and then imported to the ANSYS Design Modeler. To generate mesh for the fluid domain, ANSYS ICEM CFD was used that is a powerful tool for mesh generation which contains numerous techniques. In this tool different element shapes are available such as tetrahedral, hexahedral, prism, and pyramid with different formats such as multi-block, structured, unstructured, and many others. Because the geometry of the spillway is complex, so it is divided into some parts in order to facilitate controlling mesh type and the size. Different mesh methods were used, for sweepable parts such as chute transition and control channel sweep method was used with the manual controlling source and target face. For other parts such as stilling basin where sweep method cannot be applied multi-zone
and tetrahedron method was employed. A finer grid size was selected for those parts were high flow gradient were observed (Figure 1).

![Figure 1](image1)

**Figure 1** Typical view of mesh generation for different parts of side channel spillway.

**Boundary Conditions**

Boundary conditions have an important role in the flow simulation; accurate result can be obtained by specifying an appropriate initial and boundary conditions. ANSYS-CFX code contains several boundary conditions such as inlet, outlet, opening, wall and symmetry (Figure 2). Inlet boundary condition specified at inlet section with the average velocity, water and air volume fraction. Supercritical outlet type was specified at the outlet since the flow is supercritical, opening boundary condition was specified for the top of the spillway. Sides and bottom of the domain were specified as no slip wall boundary condition, the fluid velocity next to the wall immediately is equal to zero (ANSYS, 2011b).

![Figure 2](image2)

**Figure 2** Boundary conditions for model of side channel spillway.

**RESULTS AND DISCUSSION**

**Verification of ANSYS-CFX Results**

Because of the numerical models are based on many assumptions which influence the results, these models should be verified in order to assure accuracy of the results. The most important factors that considered in this study are meshing size and turbulence models. For this purpose the results of water surface profile obtained from ANSYS-CFX were compared with the physical model results.

**3.1.1. Mesh size**

Three models were simulated with three sets of mesh sizes, coarse, medium and fine meshes, with a number of grids (1300000, 2500000, and 3750000) for the design discharge (33.451 l/s). All models were simulated with the steady state analysis type with using k-\(\varepsilon\) turbulence model. Total computational time for fine, medium and coarse meshes were (52, 35 and 16) hours respectively. The free surface profile along the centerline and right side wall of the trough, chute channel and stilling basin were taken for comparing the results of numerical models with results of physical model (Figure 3). The percentage error between the physical model
and the ANSYS-CFX was calculated at those sections were flow depths from the physical model are measured. Finally, average percentage of error (APE) computed between ANSYS-CFX and the physical model for each part (Table 1).

### Table 1

| Parts          | APE of Grid 3.75 M with P.M. | APE of Grid 2.5 M with P.M. | APE of Grid 1.3 M with P.M. |
|----------------|------------------------------|------------------------------|------------------------------|
| Trough Channel | C.L.                         | 4.53                         | 5.23                         | 5.79                         |
|                | R.S.                         | 3.7                          | 3.82                         | 4.3                          |
| Chute Channel  | C.L.                         | 3.24                         | 3.43                         | 4.1                          |
|                | R.S.                         | 8.92                         | 9.17                         | 9.59                         |
| Stilling Basin | C.L.                         | 14                           | 15.66                        | 16.9                         |
|                | R.S.                         | 7.4                          | 13.93                        | 19.04                        |

According to the results of tables (1) the following conclusions were made:

1- The accuracy of ANSYS-CFX using 2500000 grids is almost close to that using 3750000 grids, in spite its computational time is less.

2- A noticeable deviation between the results of CFX with the physical model are exist at the end of stilling basin were strong jump occurs. This deviation due to the complexity of the phenomenon rather than mesh density.
3- For the above reasons the numerical model with 2500000 grids can be used for flow simulation in side channel spillway.

**Turbulence models**

To test the effect of turbulence models on the numerical results of water surface profile, two turbulence models were used for flow simulation in the side channel spillway. The first one is \( k-\varepsilon \) and the second is RNG \( k-\varepsilon \) turbulence model. Both simulations were performed for the design discharge (33.451 l/s). The total time required to finish both simulations with \( k-\varepsilon \) and RNG \( k-\varepsilon \) were 35, 42 hours respectively. The free surface water profile predicted by ANSYS-CFX using these two turbulence models compared with the physical model results at trough channel, chute channel and stilling basin of the spillway at the centerline and the right side wall of them (Figure 4). The APE between physical model and ANSYS-CFX are shown in Table 2.

![Figure 4 Flow depth obtained from physical model and numerical model (using k- \( \varepsilon \) and RNG k- \( \varepsilon \)).](image-url)
Table 2 The APE between physical model and ANSYS-CFX using (k-ε and RNG k-ε model) at some parts of spillway.

| Parts             | APE k-ε model with physical model | APE RNG k-ε model with physical model |
|-------------------|----------------------------------|--------------------------------------|
| Trough Channel    | C.L. 5.3                         | 5.6                                  |
|                   | R.S. 3.8                         | 3.6                                  |
| Chute Channel     | C.L. 3.44                        | 3.4                                  |
|                   | R.S. 9.2                         | 9.7                                  |
| Stilling Basin    | C.L. 15.7                        | 16.75                                |
|                   | R.S. 13.95                       | 13.4                                 |

The overall percentage of error showed that the k-ε turbulence model can be used for modeling of turbulent flow in the side channel spillway which needs less computational time than RNG k-ε model. Both turbulence models seem fail to predict water surface profile at the location of hydraulic jump. Due to high error percentages at the jump location, it needs an extra study to test the capability of k-ε turbulence model at those locations.

**Water Surface Profile (WSP)**

The results of WSP along the centerline of the spillway for various discharges predicted by ANSYS-CFX plotted with the corresponding WSP of the physical model as shown in figure 5. There is a good agreement between predicted WSP by CFX with the observed WSP from The physical model at most parts of the spillway. At the chute channel good agreement was found between CFX and physical model results especially at the centerline. While, there is differences between them at the sides of the chute channel this is mostly due to cross waves that created in the chute channel due to sharp alignment of the transition channel. The water level becomes rising and falling near boundaries of the chute channel till the flow reaches its downstream height of waves reduced (Figure 6). At stilling basin the results of ANSYS-CFX can be outlined in the following points:

1- At the beginning of the stilling basin (before jump occurs) the results of ANSYS-CFX are well agreed with results of the physical model.

2- At the jump location which characterized by high streamline curvature and high air concentration, especially for higher discharges there are differences between the flow depth obtained from CFX and that obtained from the physical model. This may be related to:
   a) The complexity of the situation and the turbulence model which cannot well detect air entrainment phenomenon. This well agree with the results obtained by (Zhan et al, 2016) (Numerical investigation of air-entrainment in skimming flow over stepped spillway) showing that RANS with k-ε is not suitable for simulation of free surface aeration. Likewise, (Castillo et al, 2014) have doubts about the suitability of using k-ε turbulence model to solve hydraulic jumps.
   b) In physical model point gauge is used for measuring flow depth which is not a good instrument for measuring water depth at the location of the hydraulic jump accurately.

3- Furthermore, as it can be observed from figures 5b, c, d, for discharges lower than the design discharge, location of the jump moves to the upstream. Differences between the physical model and ANSYS-CFX results in the stilling basin occurred at the jump location only. This deviation diminishes at very low discharges in which the hydraulic jump is unlikely form or the weak jump may be formed.
3.3. Velocity Profile

Figure 7 shows velocity distribution along the spillway obtained from CFD code CFX. The vertical velocity profile was taken at some section in the trough channel, chute channel and stilling basin as shown in figure 8, but from the physical model velocity does not measured in order to compare it with CFX. From figure 8a it is clear that the flow is non-uniform at the beginning of the trough channel. This is due to the effect of the jet of water from the crest. At the downstream sections of trough channel the effect of jet reduced, the uniform flow is nearly to achieve. In the chute channel it is clear that the flow is almost uniform, and velocity increases as flow goes downstream. Furthermore, maximum vertical velocity can be observed near the free surface flow (figure 8b). The results of vertical velocity profile in the stilling basin indicated that there is no the recirculation zone at the centerline of the stilling basin. For $X^* \ (X/L \ (\text{length of the channel})) = 0.077$ the velocity is decreased to its minimum value as depth of water increased, then it increased till reached maximum value when $Y^* \ (y/y_{\text{max}})$ was equal to 0.75. This is due to separation of incoming jet of water from bottom of the basin when spread into the stilling basin.
Figure 7 Velocity distribution with color-coded values within the spillway predicted by ANSYS-CFX

a) Trough Channel

X* = 0.148

X* = 0.296

X* = 0.444

b) Chute Channel

X* = 0.2

X* = 0.4
Flow in side channel spillway was simulated using ANSYS-CFX code which is based on finite volume method. The results of water surface profile for different discharges were compared with the physical model results. Velocity distribution predicted by CFX and vertical velocity profile was taken at various sections within the spillway. The following conclusion was established.

1. The results of k- ϵ and RNG k- ϵ turbulence models relatively close to each other at most parts of the spillway compared to the physical model results.

2. The predicted WSP by ANSYS-CFX at the center of chute channel is better agreeing with the physical model compared at the boundaries due to cross waves.

3. The ANSYS-CFX code with k- ϵ turbulence model could well predict the free surface water profile in the SC spillway components. Unlikely, it failed to give accurate predicts of WSP at the hydraulic jump when strong aeration occurred.

4. For lower discharges ANSYS-CFX code gave good predict of WSP in the stilling basin because the hydraulic jump is relatively weak.

5. Flow in the chute channel was almost close to uniform compared with the flow in trough channel and stilling basin.
REFERENCES

ABO, A. A. (2013) A three dimensional flow model for different cross section high velocity channels. PhD thesis, faculty of marine sciences and engineering, Plymouth University.

ANSYS (2011a). ANSYS CFX Release 14.0 - Theory Guide. Computer software manual.

ANSYS (2011b). ANSYS CFX Release 14.0 - Modeling Guide. Computer software manual.

AZIZ, Y. W. (2016) Evaluation of Hydraulic Performance of Nazanin Dam Side Channel Spillway. M.Sc thesis, Dams and Water Resources Department, College of Engineering, University of Salahaddin.

CASTILLO, L. G., CARRILLO, J. M., GARCIA, J. T. and VIGUERAS -RODRIGUEZ, A. (2014) Numerical Simulations and Laboratory Measurements in Hydraulic Jumps. 11th International Conference on Hydroinformatics, HIC 2014, New York City, USA, Paper 345. http://academicworks.cuny.edu/cc_conf_hic/345

GELLIBERT, A., SAVATIER, J., PEPIN, N. and FULLY, O. (2016) 3D Computational Modeling of the Galaube Dam Spillway. In Advances in Hydroinformatics (p. 361-376). Springer Singapore.

GENERAL DIRECTORATE OF DAMS AND RESERVOIRS (2013) Design Report of Nazanin Dam.

HIRT, C. W. and NICHOLS, B. D. (1981) Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics, 39 (1). P. 201-225.

LAUNDER, B. E. and SPALDING, D. B. (1974) The Numerical Computation of Turbulent Flows. Computer Methods in Applied Mechanics and Engineering, 3(2). p. 269-289.

MACHAJSKI, J. and OLEARCZYK, D. (2011) Model Investigations of Side Channel Spillway of the Zlotniki Storage Reservoir on the Kwisa River. In Experimental Methods in Hydraulic Research (p. 189-202). Springer Berlin Heidelberg.

MILESI G. and CAUSSE, S. (2014) 3D Numerical Modeling of a Side - Channel Spillway. In Advances in Hydroinformatics (p. 487-498). Springer Singapore.

PIRADEEPAN, N. (2002) An Experimental and Numerical Investigation of a Turbulent Airfoil Wake in a 90° Curved Duct. PhD thesis, Department of Mechanical Engineering, Brunel University.

SUBRAMANYA, K. (2009) Flow in Open Channels. Third edition, New Delhi, Tata McGraw-Hill Education.

TAGHZADEH, H., NEYSHABOUR, S. A.A. S. and GHASEMZADEH, F. (2012) Dynamic Pressure Fluctuations in Stepped Three - Side Spillway. Iranica Journal of Energy & Environment, 3(1). p.95-104.

VERSTEEG, H. K. and MALALASEKERA, W. (2007) An Introduction to Computational Fluid Dynamics. Second Edition, Edinburgh Gate, Harlow, Addison Wesley Longman Ltd.

YAKHOT, V., ORSZAG, S. A., THANGAM, S., GATSKI, T. B., and SPEZIALE, C. G. (1992) Development of Turbulence Models for Shear Flows by a Double Expansion Technique. Physics of Fluids A, 4(7). p.1510-1520.

ZACHOVAL, Z. and ROUSAR, L. (2015) Flow Structure in front of the Broad – Crested Weir. In EPJ Web of Conferences, 92. p. 02117. EDP Sciences. http://dx.doi.org/10.1051/epjconf/20159202117

ZHAN, J., ZHANG, J. and GONG, Y. (2016) Numerical Investigation of Air-Entrainment in Skimming Flow over Stepped Spillway. Theoretical and Applied Mechanics Letters. 6 (3). p. 139-142.