Simulating Hydraulic Characteristics and Enhance Design of My Lam Spillway-Stilling Basin,Vietnam

Le Thi Thu Hien

Abstract: The purpose of this paper is to use a hydrodynamic 3D model to simulate steady flow over spillway-stilling basin of Mylam, Vietnam. A robust and effective tool of Flow 3D software in simulating many complicated phenomena of fluid flow is selected. This model uses Volume of Fluid Method to solve Navier-Stokes equations. The computed water surface elevation at gauges along centerline of weir and chute channel are compared with experimental ones in four working conditions. The reasonable agreement between them are observed. Besides, the detail hydraulic features in each segment is indicated such as: oblique wave on spillway chute; separated flow at curve and enlarge segment and the specific hydraulic energy dissipation in stilling basin of design is carried out. The insufficient construction of dissipated obstacles in stilling basin design is shown. Therefore, a proper design is proposed.

Keywords: spillway, stilling basin, hydraulic characteristics, Flow 3D.

I. INTRODUCTION

Dam safety and ancillary facilities such as spillway, sewer, etc. always play an important role in reservoir management in Vietnam. The failure of these types of works will lead to damages, unpredictable consequences of both human and facility at downstream valley caused by the flood waves, [1,2]. Finding a proper design and verify its safety by carrying out different hydraulic features are always essential, [3]. Therefore, the study of the hydraulic characteristics of the fluid flow through these works with different operating conditions should always be considered. Mathematical models and physical models have been useful tools in studying complex hydraulic phenomena that appear in the fluid flow through irrigation works. Especially, Computational Fluid Dynamics (CFD) has been increasing dramatically in recent decades, [4,5,6,7,8]. Which can simulate precisely many hydrodynamic phenomena because they deal with Navier Stokes Equations by different numerical methods. Demeke et al (2019) used Flow 3D software to simulate the free surface flow over flood discharge system and downstream channel, [3]. While Salmasi et al (2018) used Fluent model combined with experimental study to research flow through water steps, [5].

In Vietnam, water resource development projects have been often validated carefully before construction. My Lam spillway and stilling basin is one of the most important work in Phu Yen province. Successful implementation of this project required highly attention in verifying its capability in different operating conditions. A physical model of this work was established at Laboratory of Thuyloi University. Several hydraulic features such as water depth, velocity or pressure profiles are observed. Besides, a mathematical model was also applied to simulate fluid flow over the field-scale spillway-stilling basin. Flow 3D is considered as an effective tool in studying complex hydraulic problems of flow over group of hydraulic structures, [7,8,9]. This commercial available software based on Finite Volume Method to solve Navier-Stokes equations. Therefore, in this study, the authors determine the hydraulic characteristics of the flow through flood discharge My Lam, Phu Yen also by this model. This result is compared with measured data by physical model. In addition, point out some irrationalities of the distance of the dissipated obstacles, the separation between flood flow and side walls at the bend slope, etc. of the design plan, thereby setting the plan for renovation and repair.

II. METHODOLOGY

A. PHYSICAL MODEL

My Lam spillway-stilling basin is one of the most important work in Phu Yen province, Vietnam, which includes the following main items: Ogee weir which has three 8-meter compartments; a chute steep slope channel with its bed slope of 7% and its width is 28m; a curve and enlarged segment connected this chute with a stilling basin together. There are two rows of dissipated obstruction on stilling basin and a 5m hight of wall is constructed at the end of this segment. After that, the flow goes into a large channel. Total length of this work is 145m. 1/40 scale of physical model is built to test the design plan of this project. Four working conditions of inflow and water elevations at upstream and downstream are indicated in Table 1. Besides, experimental data of water surface elevation at 10 gauges along centerline of spillway chute and curve segment are multiplied with the scale of physical model, then compared with simulated results of

Revised Manuscript Received on August 05, 2019
Le Thi Thu Hien, Thuyloi University, Hanoi, 100000 Ha noi, Vietnam.

flow over full-scale structure taken by Flow 3D model.

Fig. 1 Experimental set up of MyLam spillway and stilling basin

Table 1: Experimental test cases

N_o	Inflow Q (m^3/s)	Upstream water elevation Z_{up} (m)	Downstream water elevation Z_{down} (m)
1	800	33.48	10.68
2	1177	34.89	12.00
3	1642	36.62	13.12
4	1803	37.23	14.29

Test 2 is corresponded with design discharge, which was used to estimate dimensions of stilling basin, such as: depth, length and the height of wall at the end of stilling basin.

B. NUMERICAL MODEL

1) 3D numerical model

The momentum equations of incompressible fluid dynamics based on the second Newton law can be written as:

$$\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}=f_i-\frac{1}{\rho}\frac{\partial p}{\partial x_i}+\frac{\partial \sigma_{ij}}{\partial x_j} \quad (1)$$

where:

u_i and u_j are velocity components in x, y and z directions, respectively, $(i, j = x, y, z)$. p: pressure ; t : time

f_i : is 3 external force components exerted on a fluid particle.

σ_{ij} is turbulence stresses $\sigma_{ij}=\nu.\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)$ where ν is kinematic viscosity.

Mass conservation equation written for Newtonian, incompressible fluid in Decarse coordinate is written by:

$$\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)=0 \quad (2)$$

Both equations (1) and (2) are called Navier – Stokes equations, which is applied for incompressible fluid. In case, body force is only gravity force, Navier-Stokes equations for 4 variables (u, v, w and p) can be expressed by:

$$\begin{aligned}
&\left(\frac{\partial u_x}{\partial x}+\frac{\partial u_y}{\partial y}+\frac{\partial u_z}{\partial z}\right)=0,\\
&\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}+w\frac{\partial u}{\partial z}+\frac{1}{\rho}\frac{\partial p}{\partial x}=g_1,\\
&\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}+\frac{1}{\rho}\frac{\partial p}{\partial y}=g_2,\\
&\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}+\frac{1}{\rho}\frac{\partial p}{\partial z}=g_3.
\end{aligned} \quad (3)$$

Where g_1, g_2 and g_3 are three gravitational components in three components x, y and z.

Flow 3D software uses Volume of Fluid Method (VOF) technique for simulating many complex hydraulic phenomena. Finite volume method is used for discretization and solving each equation of RAN (Reynolds Average Navier-Stokes). The time-average process can be called Reynolds decomposition. For example, for velocity component: $u_i=\overline{u_i}+u'_i$

Therefore, Flow-3D user's manual suggests that the Renormalization Group (RNG) turbulence model is the most accurate model available in Flow 3D software in simulating free surface flow over hydraulic construction, [10].

The geometry of MyLam spillway and stilling basin is created by Auto Card 3D software and exported to Flow 3D as solid file of geometry (Fig. 2, 3). Two subdomains are divided as follow: subdomain 1includes: weir and prismatic spillway chute; subdomain 2: curve and expansion segment of spillway and stilling basin. The boundary conditions on are set up as follow: Block 1: x axis: left boundary is specific pressure; right boundary is symmetry; y axis: two side are symmetries; z axis: bottom is wall, top is specific pressure with fluid fraction is set equal 0. Block 2: x axis: left boundary is symmetry; right boundary is outflow; y axis: two side are symmetries; z axis: bottom is wall, top is specific pressure with fluid fraction is set equal 0.

Manning coefficient n is assumed 0.017. Initial conditions are water elevations at upstream and downstream as shown in Table 1. Computational domain is divided into Catersian mesh with Δx = Δy=Δz = 0.5m. Total number cells are approximate 8.10^5. In order to get steady flow in all domain, 100s is computational time, while necessary time for running program on Computer is 10 hours and result file is 7-8 GB.

Two computation modules of viscosity and turbulence, gravity are selected for all case studies in running the 3D-Euleans model. Prototype data is not available for comparison to the CFD solution, so the data from physical model is scaled to prototype dimension and considered as reference solutions.

Retrieval Number J91110881019/2019©BEIESP
DOI: 10.35940/ijitee.J9111.0881019

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

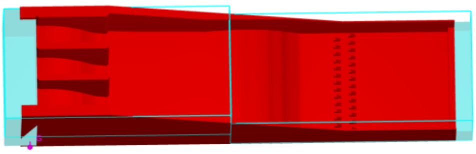

Fig 2. Model of MyLam spillway-stilling basin

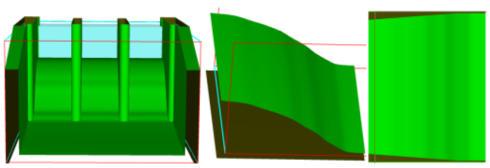

Fig 3. Weir and side view and front view of curve and enlargement segment

II. RESULTS AND DISCUSION

A. *Validations of free surface water elevation and pressure head*

The robustness and effectiveness of the CFD model in computing fluid flow over complex topography is validated by comparing predicted results of water level and piezometric pressure with observed ones. 10 gauges are set up along centerline of overflow compartment and spillway chute. Manometer is used to measure static pressure head at the bottom of gauge, while Sokia level is device to observe water elevation. 4 working conditions of inflow and initial water level in upstream and downstream are shown in Table 1.

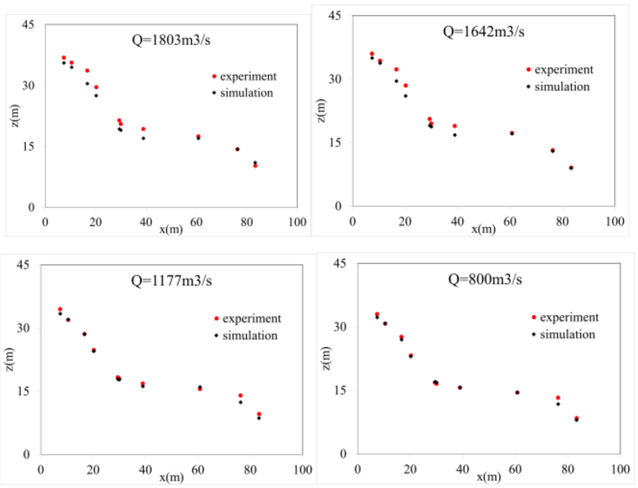

Fig. 4. Free surface water elevation in 4 operating conditions

In general, the percentage of error between physical and numerical results are in range of 5-7% in case 1 and case 2. At gauge V8, the error percentage of 4 working conditions are 2,97%; 0,7%; -2,83% and -0,14%, respectively. However, when inflow increase, the displacment between two results is larger. The piezometric pressure profiles of two later cases 3 and 4 in spillway chute is also indicated in Fig. 5. The overestimation of numerical result is seen clearly in case of Q = 1177m^3/s, but it is quite close with measured one when inflow reduces to Q = 800m^3/s. Hence, the reasonable matching between computed solution of free surface elevation and pressure head obtained by Flow 3D and impirical data is observed. Therefore, 3D software Flow 3D is quite effetive and robust in simulating the flow over groups of hydraulic structures.

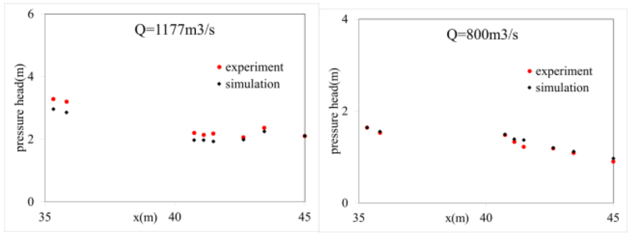

Fig. 5. Pressure head in case 3 (left) and 4 (right)

B. *Hydraulic characteristics*

1.Oblique hydraulic jump on spillway chute

Due to the impact of two pillars on the crest of weir, the oblique shock wave occurs on spillway chute, colidie each other and hit to two side walls. This fearture can be observed on both physical model (Fig. 1) and numerical one (Fig. 6). Besides, this hydraulic character also indicates that the maximum water depth and velocity do not appear at the centerline of spliiway. At water burrs, water depth is greatest while velocity is smallest (Fig. 7). Fig.7 also shows the significant difference of water depth at water burr and near side wall (8.08m compared with 0.2m).

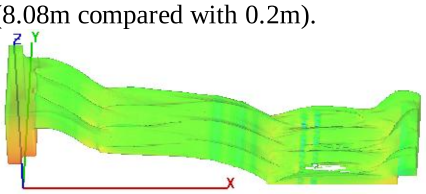

Fig. 6. Oblique shock wave on splillway

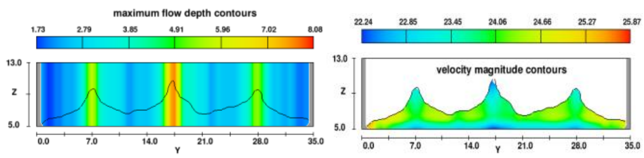

Fig. 7. Water depth and depth average velocity at x = 82.25m – Case 2: inflow Q = 1642m^3/s.

2. The separation of flow field and two side wall at curve, expanded segment.

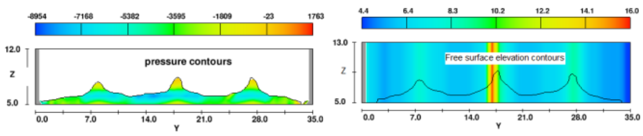

Fig. 8. Separated flow field at expanded part a) Q = 800m^3/s; b) Q = 1177m^3/s

The angle expansion of two side walls at curve part θ = 5% generates the phenomena of separation between flow field and solid wall. Especially, when volume flow rate increases, the dry part area is also increased in both sides (Fig. 8). This point show that, the enlargement of this segment is not sufficient. Hence, it should be modified the design of this part which can also optimize the width of stilling basin after.

3. Stilling basin

Stilling has two rows of dissipated energy abutments as shown in the Fig. 2. So the rapid flow from spillway chute with crash into the flow at downstream. The interference between two flows causing great turbulence. Besides, the two opposite flows also colide with two abutment rows make turbulence become greater, (Fig. 9). Therefore, the computational time needed to maintain steady flow in whole of computational domain is amost 80s (see Fig. 10).

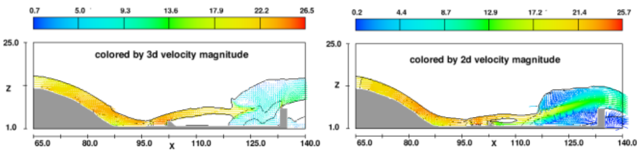

Fig. 9. The interference between upstream and downstream flows in stilling basin

C. *Modified stilling basin*

Deisgn inflow of stilling basin is 1177m^3/s, which is the most dangerous discharge level for downstream of MyLam spillway because the different water levels at upstream of weir and downstream channel is largest. The purpose of stilling basin is to dissipate energy as much as possible of very rapid flow. So its design of stilling basin as well as structure of ancillary facilities is to generate submerged hydraulic jump. In the design, there are 14 obstacles on the front row and 15 ones on the after row of dissipated aburment. The distance between them is 1.7m, which is too short and contribute to generate unsubmerge hydraulic jump (see Fig. 10) after 100s second of computational time. Therefore, authors propose a modified scenario which is increase this distance and reduce the number of obstacle to 13 and 14 in two rows of dissipated ambument. Submerged shock wave is created in new stiling basin (see Fig. 10).

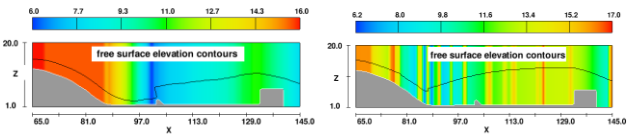

Fig. 10. Hydraulic jump in stilling basin. Design (left); Modify (right)

Besides, specific energy E is sum of water depth and velocity head should be also verified to check the sufficiency of modified plan. Fig. 11 shows that, when number of obstacle reduces, specific hydraulic head in downstream channel also decreases (10.12m compared with 8.97m) and it is contributed quite stable in cross section, meanwhile the fluctuation is seen in the design.

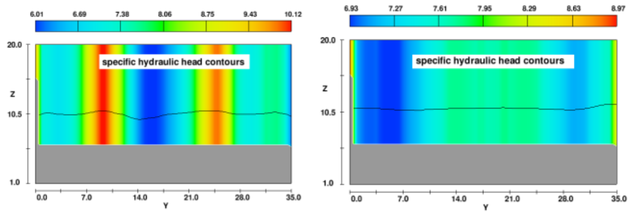

Fig. 11. Specific hydraulic head after stilling basin. Design (left); Modify (right)

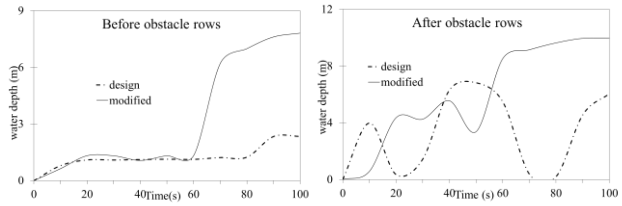

Fig. 12. Time series of water depth before and after dissipated rows

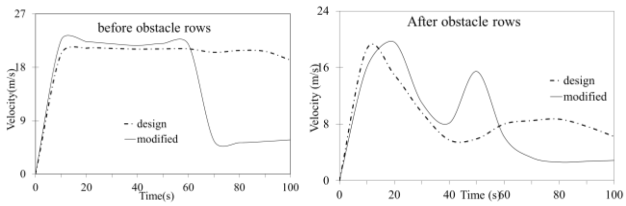

Fig. 13.Time series of velocity before and after dissipated rows.

Fig 12 and 13 indicate that, water depth before and after dissipated rows in modified case increase quickly and remain stable during 100s of computation time, while it is much smaller in the design. This point also shows that, reflect wave does not reach to obstacles rows in case of design. Besides, compare velocity at 100s before obstacle in design and modified cases, the later is much smaller than the former (5.78m/s and 19.21m/s). This trend is also seen after dissipated rows (2.87m/s and 6.25m/s). Therefore, the computed hydraulic characteristics obtained by Flow 3D model show that modified scenario is more effective and adaptable in application than the design.

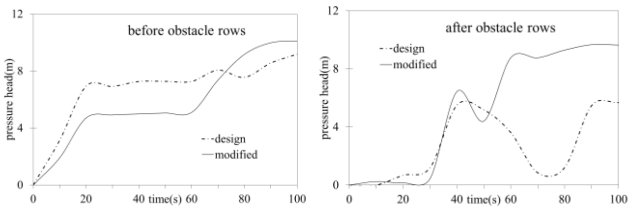

Fig. 14. Time series of piezometric pressure before and after dissipated rows

Retrieval Number J91110881019/2019©BEIESP
DOI: 10.35940/ijitee.J9111.0881019

Published By:
Blue Eyes Intelligence Engineering
& Sciences Publication

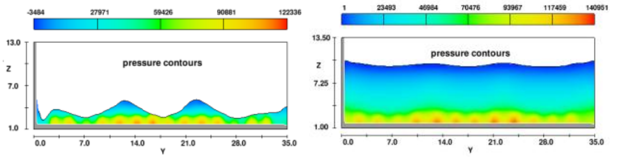

Fig. 15. Pressure distribution before dissipated rows. Design (left); Modify (right)

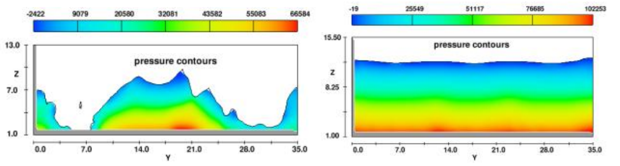

Fig. 16. Pressure distribution after dissipated rows. Design (left); Modify (right)

During computation time, piezometric pressure profiles at bottom of stilling basin before and after dissipated rows in the is much lower than that of modified case (Fig. 14). However, Fig. 15 and 16 also aim that, pressure distribution in new stilling basin is very stable in comparison with that one in the old stilling basin. This feature is important in constructing the bottom when hydrodynamic force distributes uniformly in all area.

III. CONCLUSION

My Lam flood discharge is a project with many items, so the hydraulic regime of the flow through these items is quite complicated. Flow 3D model is capable of describing the hydraulic characteristics of three dimensional turbulence on the construction accurately and intuitively. Hydraulic phenomena such as oblique waves, currents or flow interference have been described by Flow 3D. The result of water level calculated according to the mathematical model has been compared with the actual data measured in all 4 working conditions, showing a high conformity. In addition,numerical result obtained by Flow 3D software show that, repairing the stilling basin has significantly reduced the flow energy, the hydraulic features contribute more uniformly in the whole of tank.

REFERENCES

1. Serfe Yurdagul Kumcu. Investigation of flow over spillway modeling and comparison between experimental data and cfd analysis. KSCE Journal of Civil Engineering. 21(3). 994-1003. (2016).
2. G. Li, X. Li, J. Ning, Y. Deng. Numerical simulation and Engineering Application of a Dovetail-Shaped Bucket. Water, 11, 242; doi:10.3390/w11020242. (2019).
3. Getnet Kebede Demeke, Dereje Hailu Asfaw and Yilma Seleshi Shiferaw. 3D Hydrodynamic Modelling Enhances the Design of Tendaho Dam Spillway, Ethiopia. Water, 11, 82. (2019).
4. Mehmet Salih Kirkgoz , Mevlut Sami Akoz & Ahmet Alper Oner Numerical modeling of flow over a chute spillway, Journal of Hydraulic Research, 47(6), 790-797. (2009)
5. Farzin Salmasi; Aylar Samadi. Experimental and numerical simulation of flow over stepped spillway. Applied Water Science. https://doi.org/10.1007/s13201-018-0877-5, (2018).
6. Babaali, H.; Shamsai, A.; Vosoughifar, H. Copmutational Modeling of Hydraulic Jump in Stilling Basin with Convergence Wall Using CFD Cods. Arab. J. Sci. Eng., 40, 381–395 (2015).
7. Dargahi, B. Experimental study and 3D numerical simulations for a free-overow spillway", J. of Hydraul. Eng., ASCE, 132(9), pp. 899-907 (2006).
8. Zhenwei, M., Zhiyan, Z. and Tao, Z. Numerical simulation of 3-D flow field of spillway based on VOF method . Procedia Engineering, 28, pp. 808-812, (2012).
9. Le Thi Thu Hien, Do Xuan Khanh. 2D and 3D numerical evaluation of dam-break wave on an obstacle. Journal of water resources & Environmental Engineering. 62. 105-111. (2018).
10. Flow-3D User's Manual (2012). Flow science, Inc., Santa Fe, N.M.

AUTHOR PROFILE

First author: Le Thi Thu Hien – Graduated Doctor course at Brescia University – Italy since 2014. Now I am working at Division of Hydraulics, Thuyloi University, Hanoi, Vietnam.
I have experience on: Numerical modeling, Hydraulics and Hydrology. I am also involved Dam break topic; Hydraulic construction and so on.