2D AND 3D NUMERICAL EVALUATION OF DAM-BREAK WAVE ON AN OBSTACLE

Le Thi Thu Hien[1]; **Do Xuan Khanh**[1]

Abstract: *The aim of this paper is to investigate the capability of the 2D and 3D numerical models to simulate sudden dam break flows in the presence of an obstacle. The 2D-FV model is proposed based on Finite Volume method (FVM) to solve shallow water equations. The commercially-available CFD software package Flow-3D solved Navier-Stock equations along the volume of fluid (VOF) method to track the location of the free surface at the air water interface. Very good agreement of several hydraulic characteristics such as water level profile, flooding map, velocity map and force versus time history due to dam-collapse wave exerted on the obstruction produced by both presented models can be observed.*

Keywords: 2D-FV, Flow-3D, dam break wave, obstacle.

1. INTRODUCTION

Dam failures have been responsible for devastating consequences such as loss of life or destruction of property because of extreme flooding in the downstream valley. The assessment of the capability of structures to withstand flood actions is useful in emergency planning. Numerical models are useful in investigating and predicting possible flooding scenarios, which can then be used to formulate suitable flood hazard mitigation measures. Some models solved the shallow water equations (Aureli et al., 2015; Sandra Soares Frarao et al., 2008), whereas some models use 3D turbulence model (Kang et al., 2012). Much attention was paid to the hydrodynamic parameters of dam-break flows, such as velocity, discharge and free surface profile in the channel. However, the hydrodynamic forces acting on the structures which is one of the most important factor to evaluate the effect of dam-break flow to downstream structure is still ignorable. This is not because it is not important, but it is because the dam-break flow is very fast, dangerous and very difficult to measure the force accurately. Therefore, the number of studies related to force due the collision between the dam-break flow and an obstacle is still limited.

This paper aims to compare the capability of 2D shallow water, 3D Eulerian models to estimate the effect of obstacle on the dam-break wave. The 2D-FV model adopted is a finite volume method which is solved by Godunov type, second order accuracy in space and time obtained by MUSCL technique. The 3D model is the Flow 3D commercial package.

2. NUMERICAL MODEL

2.1. 2D shallow water equations

The 2D-FV model has been constructed for predicting dam break flow, which introduced in several author's works (Le, 2014, 2017). The model is used the finite volume method (FVM) for integrating the shallow water equations (SWE) numerically, MUSCL technique is applied to obtain second order accuracy in space and time. Cartesian mesh is used to generate the numerical domain.

The shallow water equations (SWEs) are derived from depth-integrating the Navier-Stokes equations and assuming hydrostatic pressure distribution. If the kinetic and turbulent viscous terms are neglected, a conservation law of the two-dimensional non-linear shallow water equations (2D-NSWE):

[1] *Division of Hydraulics, Thuyloi University*

$$\frac{\partial \mathbf{q}}{\partial t}+\frac{\partial \mathbf{f}}{\partial x}+\frac{\partial \mathbf{g}}{\partial y}=\boldsymbol{S}_o+\boldsymbol{S}_f \qquad (1)$$

$$\mathbf{q}=[h,uh,vh]^T;\ \ \mathbf{f}=\left[uh,u^2h+\frac{1}{2}gh^2,uvh\right]^T;\quad \mathbf{g}=\left[vh,\ uvh,\ v^2h+\frac{1}{2}gh^2\right]^T \qquad (2)$$

Here t is time; x and y are the Cartesian coordinates; **q** represents the flow variable vector consisting of h, uh and vh; *u* and *v* are defined as the depth-averaged velocities in *x* and *y* directions, respectively; **f** and **g** are flux vectors in x and y directions, respectively; the bed and friction slope source term $\mathbf{S}_o$ and $\mathbf{S}_f$ are expressed according to the following definitions:

$$\mathbf{S}_o=\left[0,-gh\frac{\partial z}{\partial x},-gh\frac{\partial z}{\partial y}\right]^T;\ \mathbf{S}_f=\left[0,-gh\frac{n^2u\sqrt{u^2+v^2}}{h^{\frac{4}{3}}},\ -gh\frac{n^2v\sqrt{u^2+v^2}}{h^{\frac{4}{3}}}\right]^T \qquad (3)$$

Where z denote the bed elevation and n is Manning coefficient.

With hypothesis the pressure is hydrostatic, the net force perpendicular to the vertical wall at each time step was estimated by using momentum equation applied for the control volume taken around obstacle. The total force (including hydrostatic load and momentum flux term) calculated for each cell adjacent to the two walls normal to the x-direction; in case the computational mesh is Cartesian with size Δx×Δy and the solid wall is parallel to the y-axis (along the column i), at the time level n:

$$F^n=\sum_{j_1}^{j_2}\rho g\frac{\left(h_{i,j}^n\right)^2}{2}\Delta y-\rho\frac{\left(uh_{i,j}^n\right)^2}{h_{i,j}^n}\Delta y \qquad (4)$$

The model, namely 2D-FV, based on the above numerical method is developed by the first author of this paper and validated with several test cases (Le, 2014).

2.2 3D Eulerian

In this study, a three-dimensional dam-break flow was simulated by using Flow-3D model, a powerful commercial software based on finite volume method (Flow-3D user's manual). One of the main characteristic of this dam-break flow (turbulent flow) is fluctuating velocity fields which result in the mixing of transported quantities like momentum and energy. Because of the high frequencies of the fluctuations it is difficult to simulate directly many practical problems due to their high computational cost. Therefore, the time-average equations are used instead of instantaneous equations to avoid the small scales issues and reduce the number of equations.

The time-average process can be called Reynolds decomposition. For velocity components:

$$u_i=\overline{u}_i+u_i'$$

Thus the Reynolds-averaged Navier-Stockes (RANS) equations can be expressed as:

$$\frac{\partial \overline{u}_i}{\partial x_i}=0$$
$$\frac{\partial \overline{u}_i}{\partial t}+\overline{u}_j\frac{\partial \overline{u}_i}{\partial x_j}=-\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i}+\nu\frac{\partial^2 \overline{u}_i}{\partial x_j\partial x_j}-\frac{\partial \overline{u_i'u_j'}}{\partial x_j}+g_i \qquad (5)$$

where *u* is average velocity, *p* is average pressure, *g* is gravitational acceleration

The additional term $u'_iu'_j$ are called Renolds stress and must be modelled by a turbulent model to solve the equation (5). In this study Renormalization Group (RNG) turbulent model was employed for simulation due to its high accuracy in comparison with other available turbulent model in Flow 3D such as *k*, *k* and *e* (Kermani et al., 2014).

In order to simulate the dam-break flows and their impact to an obstacle, the flow region is subdivided into rectangular cells which each cell has their own local average value of dependent variables. All boundaries of the computational domain were assumed to be rigid walls except for the top boundary (constant atmospheric pressure) and the downstream end boundary (free out flow) (Fig. 1). To estimate the force acting on the obstacle, the force is assumed to include only two terms which are hydrostatic load and momentum flux. As the results, the total force in longitudinal (*x*) direction was considered as a sum of elemental forces calculated in each cell adjacent to the two walls normal to *x* direction.

3. RESULTS AND DISCUSSION

3.1. Case study

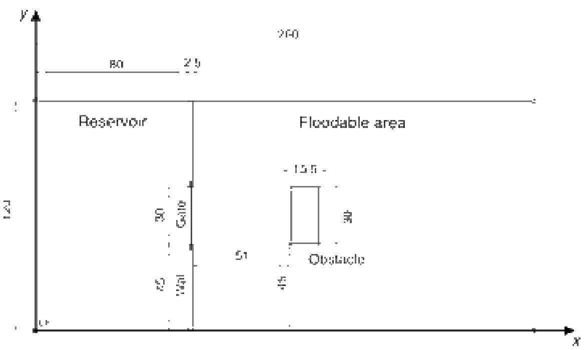

Figure 1. Configuration experimental set up

In order to evaluate the effect of an obstacle on dam-break wave by numerical method, an experiment done by Aureli et al., 2015 was used as a case study. The set-up sketched is presented in the Figure 1. The facility consisted in a 2,6m long and 1,2m wide rectangular tank divided into two compartments. The initial water depth in reservoir is 0,1m, while at the floodable area is dry. To created dam-break wave, a 0,30 m wide gage placed in the middle was set up and can be quickly opened by a simple man handle pulley system. Manning coefficient n is set equal to 0,007 and three boundaries at the upstream and at both sides are close except at the downstream is slip boundary. An obstacle is located at the center of domain.

3.2. Numerical result and discussion

In 2D-FV model, a 5mm square mesh was chosen. This mesh size was also recommended and used in the simulation done by Aureli et al., 2015 with the stability constraint was introduced by assuming the Courant number equal 0,9. Figure 2 shows simulated water depth before obstruction obtained by three model including 2D-FV model, Flow 3D and 2D shallow model performed by Aureli 2015 at t = 0,38s and t=1,44s. The formation of a hydraulic jump in front of obstacle is predicted by all three models with a maximum water depth of aproximately 7-8 cm. The water level profies before hydraulic jump obtained by 3 models were well matching. However the predicted shapes of the hydraulic jump by both two 2D models are worse than Flow 3D model due to their governing-depth averaged equations. Figure 3 presented several captured images showing the process of the flooding wave freely spreads to the downstream area simulated by 2D-FV model and Flow 3D model. There is a similarity in in both 2 models showing the collision process. In 0,38 second after the breach opening, the flood wave started reaching to the obstacle and forming an upward-moving jet in front of the wall. At 0,71s and 1,44 s the flow went around the obstacle and flooded to downstream area.

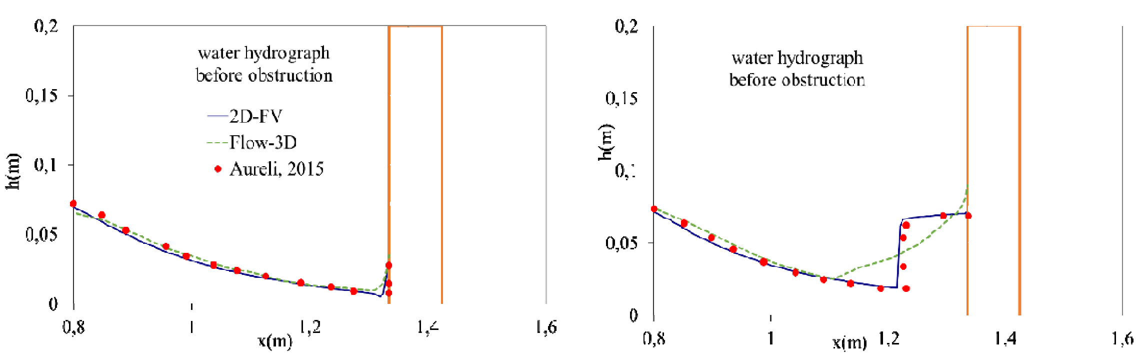

Figure 2. Water depth profiles at y=60cm before obstruction at t=0,38s and t= 1,44s.

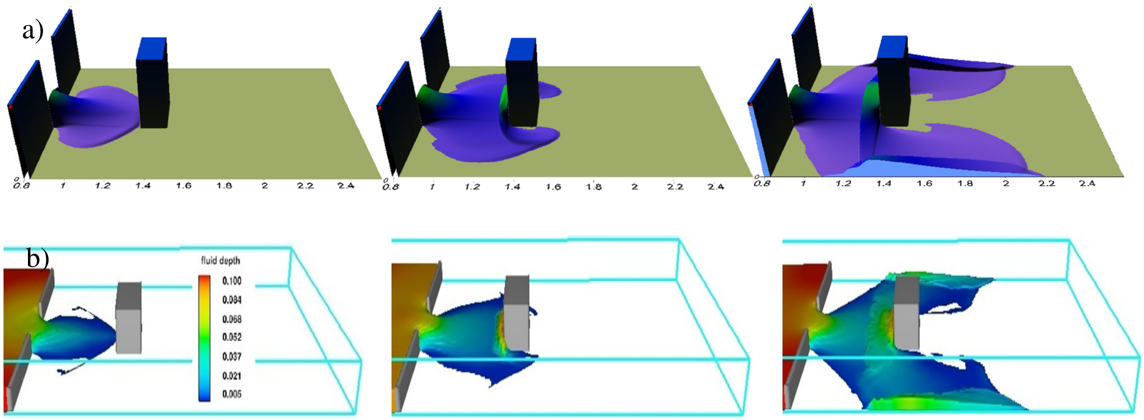

Figure 3. Flooding maps obtained by a) 2D-FV and b) Flow 3D at t = 0,38s; 0,71s and 1,44s

Figure 4 shows the velocity distribution obtained by 2D-FV model and Flow 3D in several moments. Both results indicated the maximum velocity located before hydraulic jump in front of the wall. Figure 5a is the result done by Aureli et al., 2015 presenting the comparison of the total force by time simulated by server models (2D and 3D) and experimental data. Figure 5b is the numerical load time histories simulated by 2D FV model and Flow 3D. The results indicate that both 2D models gave the similar total force profile which has only one peak with the maximum value is around 7N. The peaking time was poorly represented. It must be at around 0,7 second instead of 1,4 second. It is because of the 2D shallow approximation derives from the key assumption that vertical accelerations are negligible. Therefore, it cannot reproduce two peaks of force profile.

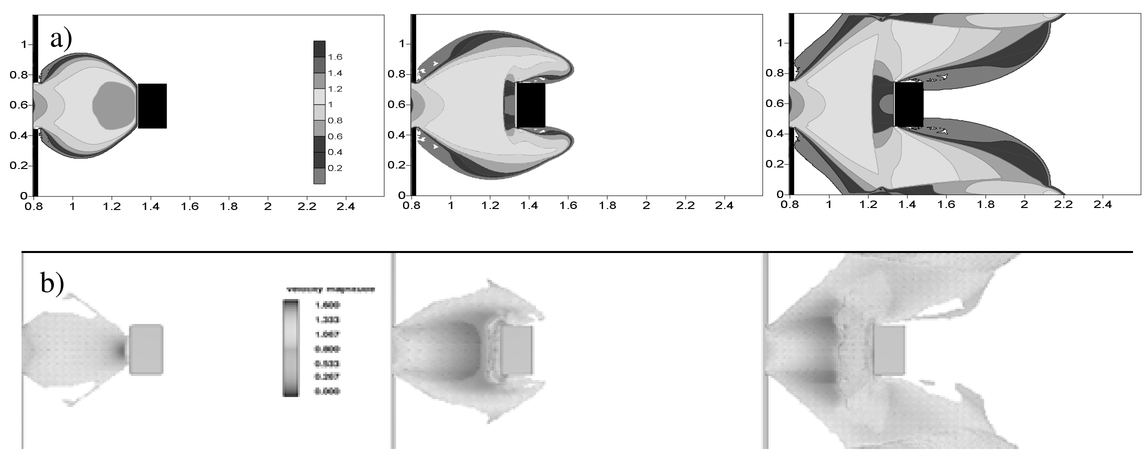

Figure 4. Magnitude velocity maps obtained by
a) 2D-FV model and b) Flow 3D at t = 0,38s; 0,71s and 1,44s.

Meanwhile, this characteristic could be well captured by all above 3D models. In comparison with solution of two 3D models in Aureli's work, numerical result of force-time history obtained by Flow-3D is much better, especially, after 1,5s. The profile of force-time is good matched with empirical solution including 2 maximum values (around 7N and 6N, respectively) and their peaking time (at 0,7 second and 1,4 second, respectively).

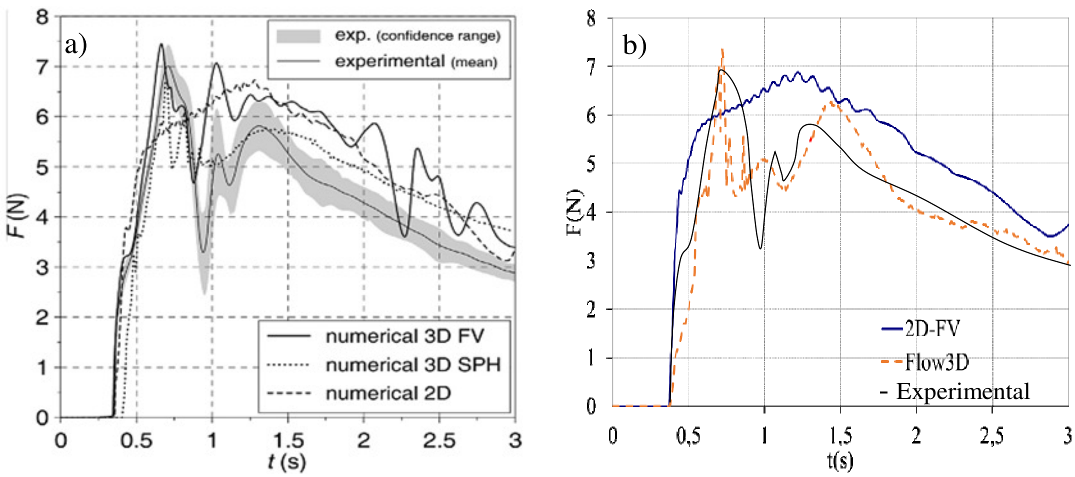

Figure 5. Comparision of load time histories simulated by several numerical model and experimental data a) Aureli et al., 2015 b) present study

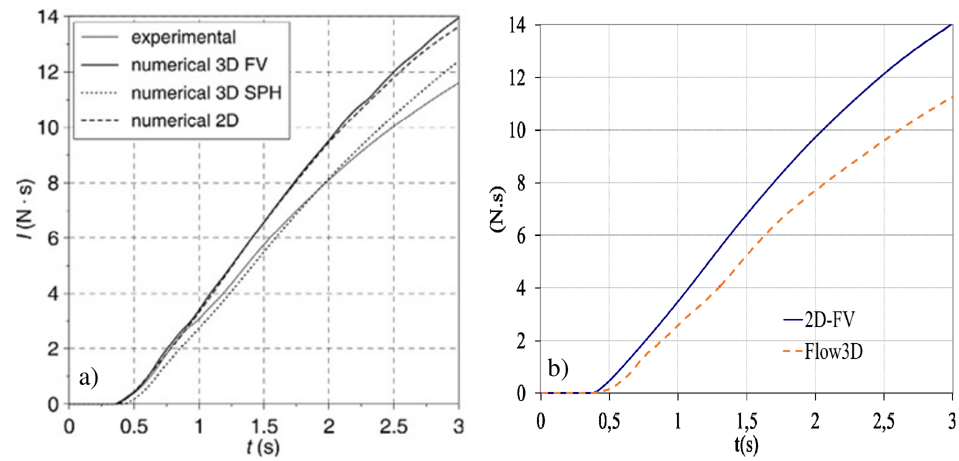

Figure 6. Comparision of load impluse time histories simulated by several numerical model and experimental data a) Aureli et al., 2015 b) present study

Figure 6 illustrates load impulse time histories. The results simulated by both 2D models (author's model and Aureli) are similar. Both of them, however, are over estimated in comparing with experimental data. In the time of 3 second, the difference between them is around 2,5N. In addition, all above 3D models except Flow 3D also are over estimated, however their differences are much smaller. Flow 3D presents as the most suitable model in simulating force acting on an obstacle. The accumulated total force at 3 second is well-matching with the observed data which is approximately 11,8N.

4. CONCLUSION

The achievement of this paper is used both 2D and 3D models to reproduce a case study of dam break flow acting to an obstruction. An 2D-FV numerical model was proposed based on FVM with high order accuracy in space and time to solve SWEs and a commercial software Flow-3D is applied to obtain several hydraulic characteristics such as: water depth profile, inundation maps of water depth and magnitude of velocity. Force time and load impulse time history are also calculated by both selected model. The results show that, there is a good agreement between water depth and velocity simulated by both proposed models. Unfortunately, double-peak trend of force time relation could not be predicted by2D-FV model. However, it still can estimate the maximum value of the total force acting on an obstacle. Meanwhile, numerical simulation of force time history and load impulse time history indicated that Flow-3D is the most suitable model. It can simulate more closely to experiment data published by Aureli et al., 2015 in comparison with the other 3D model. In next study, the impact of group of obstacle on dam break flow will be considered.

REFERENCE

F. Aureli, A. Dazzi, A. Maranzoni, P. Mignosa, R. Vacondio (2015). *"Experimental and numerical evaluation of the force due to the impact of a dam break wave on a structure"*. Advances in Water Resources, 76, 29-42.

Sandra Soares-Frazao; Yves Zech (2008). *"Dam-break flow through an idealized city"*. J. Hydraulic Research, 46(5), 648–65.

S. Kang, F. Sotiropoulos (2012). *Numerical modeling of 3D turbulent free surface flow in natural waterways.* Advances in Water Resources, 40,23-36

Le T.T.H (2014). *"2D Numerical modeling of dam break flows with application to case studies in Vietnam"*, Ph.D thesis, University of Brescia, Italia.

Le Thi Thu Hien, Vu Minh Cuong (2017). *"Studying an efficient second order accurate scheme for solving two-dimensional shallow flow model"*, Tạp chí Thủy lợi và Môi trường, 60, 117-124.

Kermani, E. F. and Barani, G. A (2014). *"Numerical simulation of flow over spillway based on CFD method"* Scientia Iranica A, 21(1), 91-97

Tóm tắt:

NGHIÊN CỨU MÔ HÌNH TOÁN HAI CHIỀU VÀ BA CHIỀU TRONG ĐÁNH GIÁ ẢNH HƯỞNG CỦA VẬT CẢN TỚI SỰ LAN TRUYỀN SÓNG VỠ ĐẬP

Bài báo này nghiên cứu khả năng của mô hình toán 2 chiều và 3 chiều trong mô phỏng dòng chảy lũ chịu ảnh hưởng của vật cản. Mô hình 2 chiều 2D-FV do tác giả xây dựng dựa trên phương pháp thể tích hữu hạn để giải hệ phương trình nước nông hai chiều. Mô hình thương mại 3 chiều Flow-3D dựa trên phương pháp VOF để giải hệ phương trình Navier-Stokes cũng được sử dụng. Những kết quả tính toán như quá trình mực nước, bản đồ ngập lụt về độ sâu, lưu tốc hay lực tác dụng bằng hai phương pháp trên được phân tích so sánh với kết quả theo phương pháp số hay thực nghiệm của Aureli và nnk, 2015 cho thấy sự phù hợp cao.

Từ khóa: 2D-FV; Flow-3D, dòng chảy do vỡ đập, vật cản.

Ngày nhận bài: 30/5/2018

Ngày chấp nhận đăng: 08/8/2018