

## **Numerical and Experimental Investigation of Dam-Break Wave on a Single Building Situated Downstream**

Selahattin Kocaman<sup>1</sup>, Hasan Güzel<sup>1</sup>

<sup>1</sup>*Department of Civil Engineering, Mustafa Kemal University, İskenderun, Turkey*

### **ABSTRACT**

Dam-break flow may cause severe damages on land and population located downstream due to flush flooding. In addition to the complexity inherent in dam break flow, existence of buildings changes the rapidly varying flow pattern downstream making the problem more complex. The way the buildings are oriented and clustered has a great influence on water depths and velocities of the dam-break flow. To understand the phenomena better, in this study, the flow around a single building located vertically and excentred with respect to the main flow direction are investigated experimentally and numerically. The wave propagation and its interaction with the building was observed using high speed camera and obtained results were compared with FLOW-3D numerical model which uses 3D RANS equations utilizing k-ε turbulence model. The results were in good agreement.

**Keywords:** *Dam-break, unsteady flows, obstacle, FLOW-3D, CFD*

### **INTRODUCTION**

The propagation of a dam-break generated flood wave may have catastrophic effects downstream due to sudden release of large amount of water from the reservoir. The prediction of the free surface form, water depths and flood wave location is significant for reducing losses of lives and flood damages. The irregular topography and presence of man made obstacles such as buildings, bridges and roads play an important role on the maximum flow depth, flood-wave propagation velocities and flow regime at downstream of a dam. the orientation of such obstacles with respect to main flow direction also influences the flow behavior and complexity of the inherently turbulent flow. In addition, dam-break flows exert significant impact force on structures or obstacles due to rapid propagation speed.

There are limited numbers of experimental investigations describing two- and three-dimensional dam-break flows due to the difficulty in measuring dynamic unsteady open channel flow parameters. Some of these studies were conducted a horizontal rectangular channel for dry and wet conditions in which the dam-break was represented by sudden removal of a vertical plate [1-3]. The others studied dam-break flow propagation over irregular topography having sudden bottom variation and contraction at downstream channel [4-6]. For 3D cases, few studies are available especially involving presence of a building against dam-break flood wave [7-8].

In recent years, dam-break flow studies have been mainly directed at numerical solution because of difficulties in getting field data. Numerical models based on shallow-water equations (SWEs) are mostly used to describe unsteady dam-break flows due to the advantage of requiring considerably less CPU time for large computational domains [9-11]. In these

flows, there appear complicated multiple wave interactions and strong discontinuities in the free surface. SWE solutions, which have some limitations, can not accurately predict these flows due to hydrostatic pressure assumption. Therefore, Computational fluid dynamics (CFD) models based on the Reynolds Averaged Navier Stokes (RANS) equations with turbulence closure have recently become useful tools in solving such complex rapidly-varied and unsteady flow problems [12-14].

The purpose in the present study was to investigate the dam-break flow experimentally and numerically in the presence of a single building located vertically and excentred with respect to the main flow direction in the downstream channel. The building acts as an obstacle against the flow which influences the pattern and the dynamics of the flow. With this aim, laboratory experiments were conducted to observe dam-break event by using high speed camera recording the behavior and then compared with the numerical results. The motivation was to show the capability of the CFD based FLOW-3D software in representing the dam-break phenomena in terms of the direction and the interaction of the flood wave in the presence of an obstacle, namely a building. In numerical simulations,  $k-\epsilon$  turbulence model was used.

## **LABORATORY SETUP**

The experiments were carried out using a rectangular horizontal channel 1.00 m long, 0.50 m wide, and 0.35 m high (Figure 1a). The channel bottom and walls were made of 9 mm thick glass. The dimensions of water body were 0.25m long and 0.50m wide representing the reservoir. The initial reservoir water depth was constant at  $h_0 = 0.20$  m. The plate was located at the longitudinal center of the channel representing a dam. The 4 mm thick plate, coated with aluminium, was made of rigid plastic.

To model dam-break flow, a removal mechanism was constructed to be able to remove the vertical plate instantaneously (Figure 1b). A steel rope was connected to the plate top; the rope was drawn over a pulley with 15 kg weight hanging at the other end. By releasing the weight from 1.0 m above the floor, the plate was removed. The plate removal time was between 0.06 s from video records. Lauber and Hager [1] recommended that it be shorter than  $1.25(h_0/g)^{1/2}$  for a 'sudden removal', corresponding to 0.18 s in the present tests. Here,  $h_0$  is reservoir depth and  $g$  is acceleration due to gravity.

At test beginning, the reservoir was filled with water, then the plate was lifted after the still water condition was achieved. Grease oil was used to prevent the leakage from the contact edges between the plate and the channel walls. Downstream part of the channel was dried carefully to achieve dry bed conditions. An obstacle, namely building, with the dimensions of 8cm x 15cm was located at 30 cm downstream from the plate. A total of two tests were conducted. The obstacle was placed vertically and rotated with respect to the main flow direction in test-1 and test-2, respectively (Figure 1a). In both tests, the flood wave propagated over a dry bed.

To identify the behavior of dam-break flow via high speed camera, the reservoir water was coloured using dye. A high speed CCD camera having 300 fps was used to better observe the dam-break flow which happened in a quite short time of 2 seconds in present tests. The test area was lightened by fluorescent spotlight located above the channel. The camera was placed right at the top of the building to be able to see whole downstream area and observe flow behavior symmetrically.

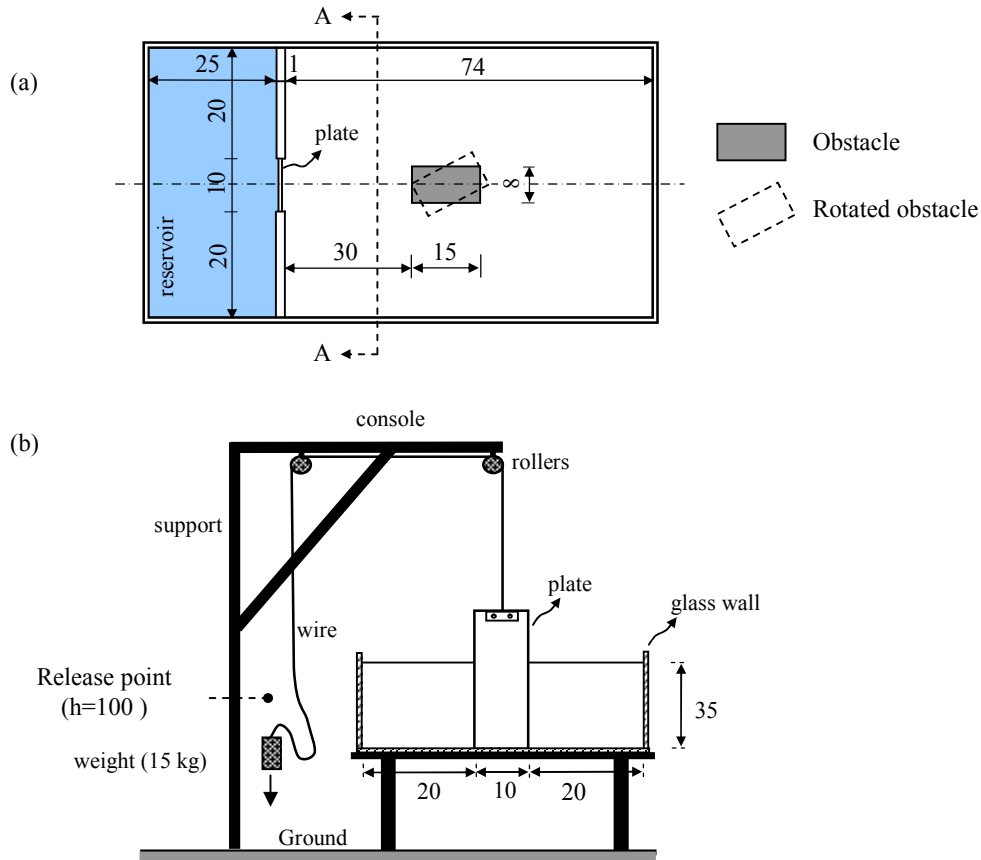


Figure 1 Schematic view of experimental setup and plate removal mechanism  
 a) plan b) A-A cross-section, lengths in [cm]

## NUMERICAL MODEL

With the computing advancements, CFD has emerged as a powerful hydraulic design tool. The commercially-available CFD program FLOW-3D developed by Flow Science Inc., Los Alamos NM was used herein. FLOW-3D was designed to treat time-dependent flow problems in one, two and three dimensions. It is claimed to be applicable to almost any type of flow and provides many options for users including explicit and implicit solutions and five different turbulence closure approaches: Prandtl's mixing length, one-equation turbulence energy, two-equations  $k-\varepsilon$  equation, ReNormalization-Group (RNG), and large eddy simulation. For 3D free surface flow simulation, the Reynolds-Averaged Navier-Stokes (RANS) equations are solved [15].

The geometrical specifications are independently applied on a grid, enabling the generation of complex obstacles. Each obstacle considered can be implemented in the flow domain. The program evaluates the location of the flow obstacles by utilizing a cell porosity technique called Fractional Area/Volume Obstacle Representation (FAVOR) method. An obstacle in a mesh is thus represented by a volume fraction (porosity) value ranging from zero to one as the obstacle fills in the mesh. Hence, for an obstacle the following conditions are distinguished: completely solid, partly solid and fluid, completely fluid, partly fluid, or completely empty. In FLOW-3D, the Volume-Of-Fluid (VOF) technique was used to model free surfaces [15].

## APPLICATION OF FLOW-3D

The computational domain was subdivided into a mesh of fixed rectangular cells using Cartesian coordinates. Prior to computation, the computational domain, 1.00 m long, 0.50 m wide, and 0.20 m high was divided into structured grids, and boundary and initial conditions were prescribed (Figure 2). The computational domain involved uniform rectangular grids spaced as  $\Delta x = \Delta y = \Delta z = 0.05$  m, resulting in 200, 100 and 40 cells in the x, y and z directions, respectively. The total number of cells was therefore 800,000.

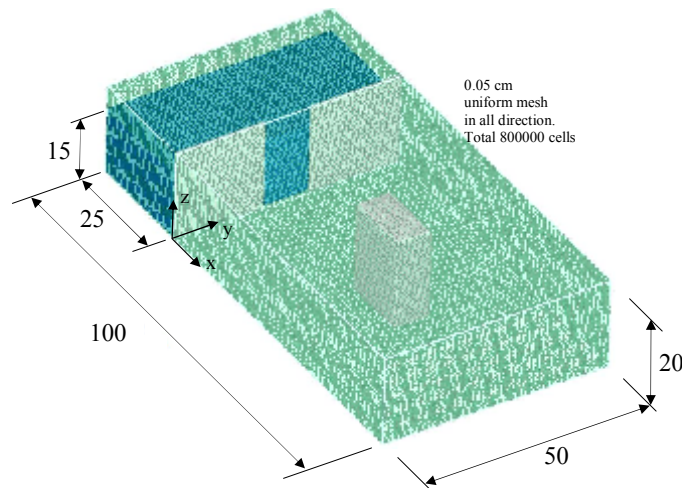


Figure 2 3D configuration of numerical model

To represent the physical flow conditions accurately, the boundary conditions have to be carefully defined. In the numerical computations, the sidewalls as well as the channel bottom were set as wall due to no flow into the reservoir. At the top, the symmetry (atmospheric) boundary condition was assigned to account for the atmospheric pressure on the free surface. Since the water surface is defined by VOF, zero shear stress and constant atmospheric pressure were applied as boundary conditions over the air-water interface [15]. A constant volume of fluid with dimensions of 25x50x20cm representing the reservoir was assigned as an initial condition (Figure 2). For turbulence closure, the k- $\epsilon$  method was applied. All channel and obstacle surfaces were assumed smooth. The no slip condition was defined as zero tangential and normal velocities.

Once the initial and boundary conditions were established, the model was applied for two different obstacle orientations. Simulation was run for  $t=2$ s. The time step  $\Delta t$  was determined according to the Courant-Friedrichs-Lewy condition.

## RESULTS

As can be seen in Figures 3 and 4, after sudden removal of the plate, strong dam-break wave propagates in downstream direction. As the wave reaches the building it reflects against the building increasing the flow depth locally on upstream side confronting the wave and then flow separates. The wave front also reflects against the channel side walls and oblique hydraulic jumps are formed. At the initial and following stages of dam-break flow, wake zone occurs behind the building. Re-circulation zones can also be observed between the building

and the walls. The separated flow moving around the building and jumps against channel side walls are merged at downstream end, then, reflecting from the downstream wall resulting in negative waves. These negative waves move upstream towards the building and overflow the wake zone.

Aforementioned observations indicate that the flow in downstream channel is potentially very complex, involving multiple wave interactions, hence requiring a demanding test for numerical methods. In the present study, RANS based numerical method was compared with experimental results to test the capability of FLOW-3D in simulating such a complex unsteady flow problem. The comparison of the results showed good agreement (Figures 3 and 4). FLOW-3D could simulate with reasonable accuracy the behavior of the flow at the initial and further stages of dam break event in presence of obstacles.

In the experiments, it was observed that flow acts in a different way when the building is placed vertically (test-1) and excentred (test-2) relative to the main flow direction creating larger wake zone ( $t=0.7s$ ) and increasing flow circulations ( $t=1.2s$ ) behind the building in the later case. FLOW-3D also captured these flow patterns for both tests fairly except just downstream from the building at the edges of the wake zone ( $t=0.7-1.2s$ ) where the flow depth is low in test-2. This may be overcome by using better mesh resolution.

In test-1, building was exposed to the flow on the front and on both sides (Figure 3) whereas in test-2 it was exposed to the flow on two sides and also guided the flow away by acting as a wedge resulting in larger wake zone behind the building (Figure 4). This wedge-like effect also directs the flow, which confronts the narrow side of the building, upward higher. This difference in exposure and building-flow interaction may induce higher forces due to impact on the front face and higher drag on the building in test-1 compared to test-2.

## **CONCLUSION**

This work presents influence of the presence of a single building representing irregular topography on dam-break flood wave propagation over horizontal dry bed for practical applications. The study was carried out on a single building located vertically and excentred relative to the main flow direction.

When the dam-break flood wave encounters the obstacle, the pattern and the route of the flow around it changes due to flow concentration in front of the obstacle and then accelerates on both sides. This results in complex flow. Thus, It is important to obtain the water depths and flow directions in this kind of dam-break flow. However, it is a very difficult task, especially in the field, because of the rapidly-varied and unsteady nature of the dam-break flows. A high speed camera was used to obtain a good overview of the flow features. The acquired experimental data was compared with the RANS based FLOW-3D numerical simulation software utilizing  $k-\epsilon$  turbulence model. Despite some discrepancies, there was good agreement between experimental and computed results for both tests. Just downstream from the building, a larger wake zone was observed for excentred building compared to vertically located one. The results indicated that FLOW-3D can be a useful tool in simulating dam-break flow which lacks sufficient field data.

Acquisition of higher quality laboratory data especially consisting of water depth and pressure measurement can be useful for numerical researchers to validate their models concerning dam-break flood propagation over complex topography.

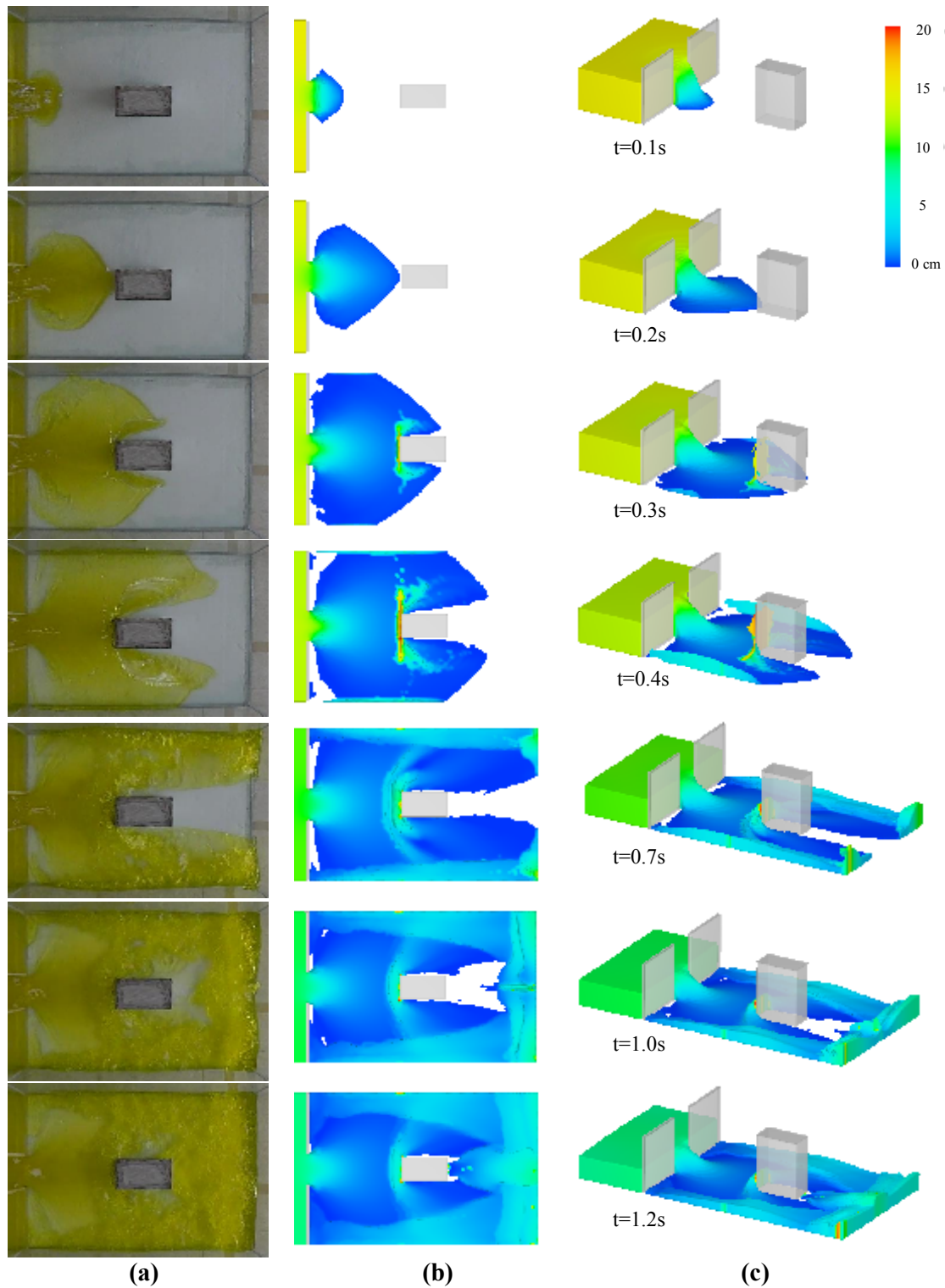


Figure 3 Comparison of a) experimental b) numerical c) 3D numerical results at different times for obstacle placed vertically relative to the flow direction



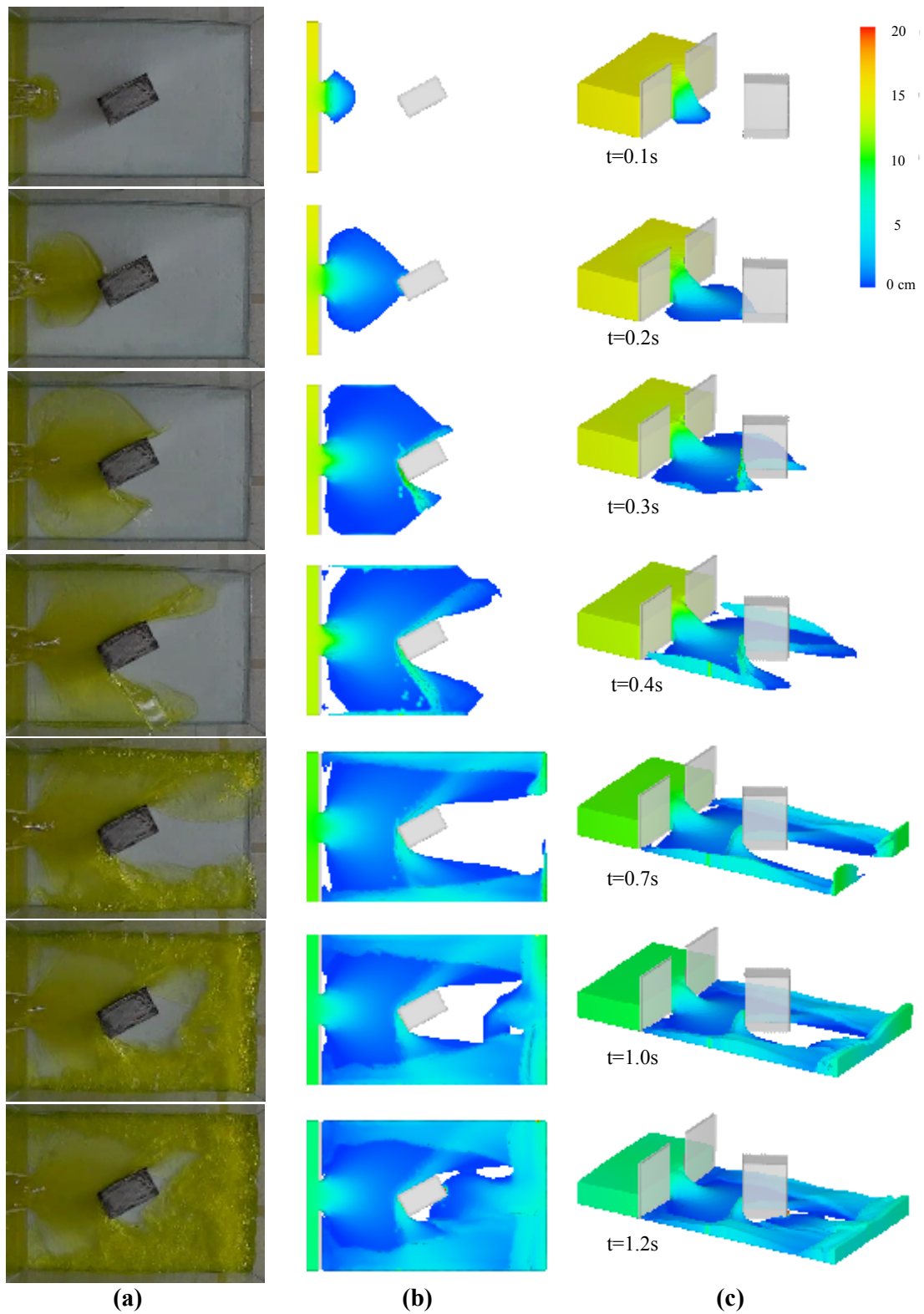


Figure 4 Comparison of a) experimental b) numerical c) 3D numerical results at different times for obstacle excentred relative to the flow direction

## **REFERENCES**

- [1] Lauber, G., Hager, W.H., (1998) Experiments to dam-break wave: horizontal channel. *Journal of Hydraulic Research*, 36(3), 291-308.
- [2] Stansby, P. K., Chegini, A., Barnes, T.C.D., (1998) The initial stages of dam-break flow. *J. Fluid Mech.*, 374, 407-424.
- [3] Cagatay, H., Kocaman, S., (2008) Experimental study of tail water level effects on dam-break flood wave propagation. *River Flow 2008, Proc., of the International Conference on Fluvial Hydraulics, Cesme, Izmir, Turkey, Vol.1*, 635-644.
- [4] Soares-Frazao S. (2007) Experiments of dam-break wave over a triangular bottom sill. *Journal of Hydraulic Research*, 45 (Extra Issue), 19-26.
- [5] Aureli F, Mignosa P, Tomirotti M (1999) Dam-break flows in presence of abrupt bottom variations. *Proceedings of XXVIII IAHR Cong 1999, Graz, Australia*, 163-171.
- [6] Bellos, V., Soulis, J.V., Sakkas, J.G., (1992) Experimental investigation of two-dimensional dam-break induced flows. *Journal of Hydraulic Research*, 30(1), 47-63.
- [7] Soares-Frazao, S., Zech, Y. (2007) Experimental study of dam-break flow against an isolated obstacle. *J. Hydr. Res.*, 45 (Extra Issue), 27-36.
- [8] Aureli, F., Maranzoni, A., Mignosa, P., Ziveri, C., (2008) Dam-break flows: Acquisition of experimental data through an imaging technique and 2D numerical modelling. *ASCE J. Hydr. Engng.*, 134(8), 1089-1101.
- [9] Brufau P., Garcia-Navarro P., (2000) Two-dimensional dam-break flow simulation. *Int J Numer Methods Fluids*, 33(1), 35-57.
- [10] Liang Q, Marche F, (2009). Numerical resolution of well-balanced shallow water equations with complex source terms. *Advances in Water Resources*, 32(6), 873-884.
- [11] Quecedo, M., Pastor, M., Herreros, M.I., Merodo, J.A.F., Zhang, Q., (2005) Comparison of two mathematical models for solving the dam break problem using the FEM method. *Comput. Methods Appl. Mech. Engrg.*, 194, 3984-4005.
- [12] Shigematsu, T., Liu P.L.F., Oda, K., (2004) Numerical modelling of the initial stages of dam-break waves. *Journal of Hydraulic Research*, 42(2), 183-195.
- [13] Biscarini C., Francesco D.S. and Manciola P. (2010) CFD modelling approach for dam break flow studies. *Hydrology and Earth System Sciences*, 14(4), 705-718.
- [14] Ozmen-Cagatay H. and Kocaman S. (2010) Dam-break flows during initial stage using SWE and RANS approaches. *Journal of Hydraulic Research*, 48(5), 603-611.
- [15] FLOW SCIENCE Inc., (2010) *Flow-3D User's Manuals*, Santa Fe, N.M.