# Three-dimensional numerical analysis of a dam-break using OpenFOAM

[1, 2] *Esteban Sánchez-Cordero <esteban.sanchezc@ucuenca.edu.ec>*
[2] *Manuel Gómez <manuel.gomez@upc.edu>*
[2] *Ernest Bladé <ernest.blade@upc.edu>*
[1] *Department of Civil Engineering, Faculty of Engineering,*
*University of Cuenca, Cuenca, Ecuador*
[2] *Department of Civil and Environmental Engineering,*
*FLUMEN Research Institute,*
*Polytechnic University of Catalonia, Barcelona, Spain*

**Abstract**. This paper presents a 3D numerical analysis of flow field patterns in a dam break laboratory scale by applying the numerical code based on Finite Volume Method (FVM), OpenFOAM. In the numerical model the turbulence is treated with RANS methodology and the VOF (Volume of Fluid) method is used to capture the free surface of the water. The numerical results of the code are assessed against experimental data. Water depth and pressure measures are used to validate the numerical model. The results demonstrate that the 3D numerical code satisfactorily reproduce the temporal variation of these variables.

**Keywords**: dam break; 3D; RANS; k-ε (RNG); OpenFOAM

## 1. Introduction

A dam is an engineering structure constructed across a valley or natural depression to create a water storage reservoir. The fast-moving flood wave caused by a dam failure can result in the loss of human lives, great amount of property damage, and have a severe environmental impact. Therefore, significant efforts have been carried out over the last years to obtain satisfactory mathematical numerical solutions for this problem. Due to advances in computational power and the associated reduction in computational time, three-dimensional (3D) numerical models based on Navier Stokes equations have become a feasible tool to analyze the flow pattern in those days.

Analytical studies of the dam break for a horizontal channel were performed by Dressler [1]. Several numerical studies based on 2D approaches have been validated against experimental data sets as demonstrated in [2] and [3]. Two dimensional numerical models assume negligible vertical velocities and accelerations which results in a hydrostatic pressure distribution. However, when an abrupt failure of a dam happens, in which initially a high free surface gradients occurs, the hydrostatic pressure assumption is no longer valid. Three dimensional numerical models have been used to solve the structure of the flow in these areas.

This document presents a 3D numerical analyze of a dam break (laboratory scale) using the numerical code based on the finite volume method (FVM) – OpenFOAM. Turbulence is treated using Reynolds-averaged Navier Stokes equations (RANS) k-ε (RNG) approach, and the volume of fluid (VOF) method is used to simulate the air-water interface. The numerical results of the code are assessed against experimental data obtained by Kleefsman et al [4]. Water depth and pressure measures are used to validate the model. The results demonstrate that the 3D numerical code satisfactorily reproduce the temporal variation of these variables.

## 2. Experimental set-up model

Fig. 1 shows the schematic and the positions of the measured laboratory quantities performed by Kleefsman et al [4]. The dimensions of the tank are 3.22x1x1 m. The right part of the tank can be filled up with water up a height of 0.55 m using a sluice gate. Then the sluice gate opens abruptly and the water in the tank empties.
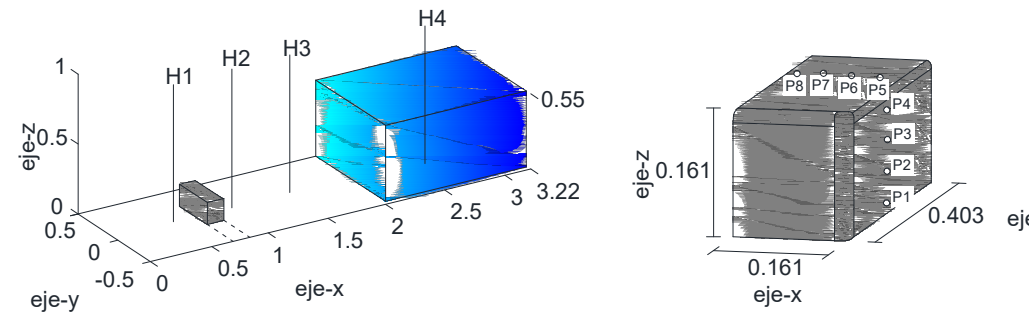


Fig. 1. Measurements positions for water heights and pressure (adapted from Kleefsman et al [4])

Санчес-Кордеро Э., Гомез М., Блейд Э. Трехмерный численный анализ прорыва плотины с использованием OpenFOAM *Труды ИСП РАН*, том 29, вып. 6, 2017 г., стр. 311-320

Sánchez-Cordero E., Gómez M., Bladé E. Three-dimensional numerical analysis of a dam-break using OpenFOAM. *Trudy ISP RAN/Proc*. ISP RAS, vol. 29, issue 6, 2017. pp. 311-320

## 3. Numerical Model

### 3.1 Fluid Flow model

The governing equations for mass and momentum for the fluid flow can be expressed as [5]:

$$\nabla \cdot u = 0 \qquad (1)$$

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u\, u) - \nabla \cdot \big((\mu + \mu_t)S\big) = -\nabla p + \rho g + \sigma K \frac{\nabla \alpha}{|\nabla \alpha|} \qquad (2)$$

where u is velocity vector field, $p$ is the pressure field, $\mu_t$ is the turbulent eddy viscosity, $S$ strain tensor ($S = 1/2\,(\nabla u + \nabla u^t)$ , $\sigma$ surface tension, $K$ surface curvature, and $\alpha$ volume fraction function (between 0-1).

### 3.2 Free surface model

Volume of Fluid Method (VOF) is used for the analysis of free surface flow. A volume fraction indicator $\alpha$ is used to determine the fluid contained at each mesh element. To calculate $\alpha$ a new transport equation is introduced.

$$\frac{\partial \alpha}{\partial t} + \nabla(\alpha \cdot u) + \nabla \cdot (\alpha(1-\alpha)u_r) = 0 \qquad (3)$$

OpenFOAM imposes the third term of equation (3) called phase compression; where, $u_r = u_l - u_g$. The density $\rho$ and viscosity $\mu$ in the domain are given by:

$$\rho = \alpha \rho_l + (1-\alpha)\rho_g \qquad (4)$$

$$\mu = \alpha \mu_l + (1-\alpha)\mu_g \qquad (5)$$

where $l$ and $g$ denotes the different fluids (water and air).

### 3.3 Turbulence model

In the RANS equations the instantaneous variables of flow are decomposed into their time-averaged and fluctuating quantities. In this analysis k-ε (RNG) turbulence model is used due to provides an improved performance for types of flows that include flows with separation zones [6]. It is a two equation model which provides independent transport equations for both the turbulence length scale and the turbulent kinetic energy.

### 3.3 Initial and Boundary Conditions

In the numerical configuration of the model, the sides surrounding the experiment and the bottom are defined as wall. The top of the experimental box atmospheric pressure prevails. At the beginning of the simulation an initial water height is established, which is the initial water volume of the experiment.

### 3.4 Model validation – Grid convergence

In this subsection, three grid resolution values are evaluated for the grid convergence. The domain is discretized using a structured mesh made up of hexahedral elements. The mesh sizes to be analyzed are 2, 1.5 and 1 cm. The water depth variable is chosen as the analysis due to the reliability of the measurement of its values. In order to quantify the numerical assessment quadratic mean value $R^2$ is used. Fig. 2 shows how the statistical value $R^2$ increases when the mesh size decreases in the four measurement points.
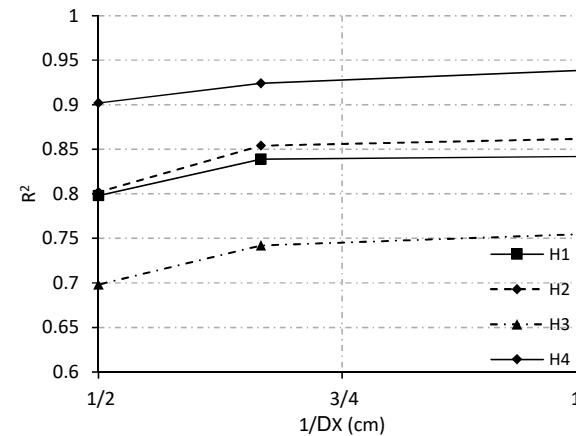


Fig. 2. Grid mesh analysis

### 3.5 Numerical Simulation

In this study, after the mesh analysis a grid of 1 cm is used. The grid cells has been used with some narrowing towards the bottom and the walls of the tank. An Explicit 2nd order limited scheme for the convection term, Explicit second order scheme for the diffusion term, and first order Euler scheme for the transient term are used. The simulation is continued for 7 s with an automatically adapted time step using maximum CFL-numbers around 0.50.

## 4. Results and Discussion

In order to analyze the capabilities of the numerical model in the reproduction of the flow variables in a dam break, a comparison between RANS numerical simulation and experimental data was quantified by the quadratic mean value $R^2$.

### 4.1 Water Depth

The evolution in time of water depth (H1, H2, H3, and H4) are shown in Fig. 3. A qualitative evaluation of the results shows that the 3D model configuration is able to reproduce satisfactorily the variability in time of the water depth in the four points of study. Additionally, a qualitative evaluation of the results shows that the 3D numerical model explains the variability in time of water depth (Fig. 4). The best numerical data occurs at the point denominated H4 (Fig. 4-d), this point is located inside the water tank formed at the beginning of the experiment. On the other hand, the worse numerical data occurs in the point denominated H3 (Fig. 4-c).
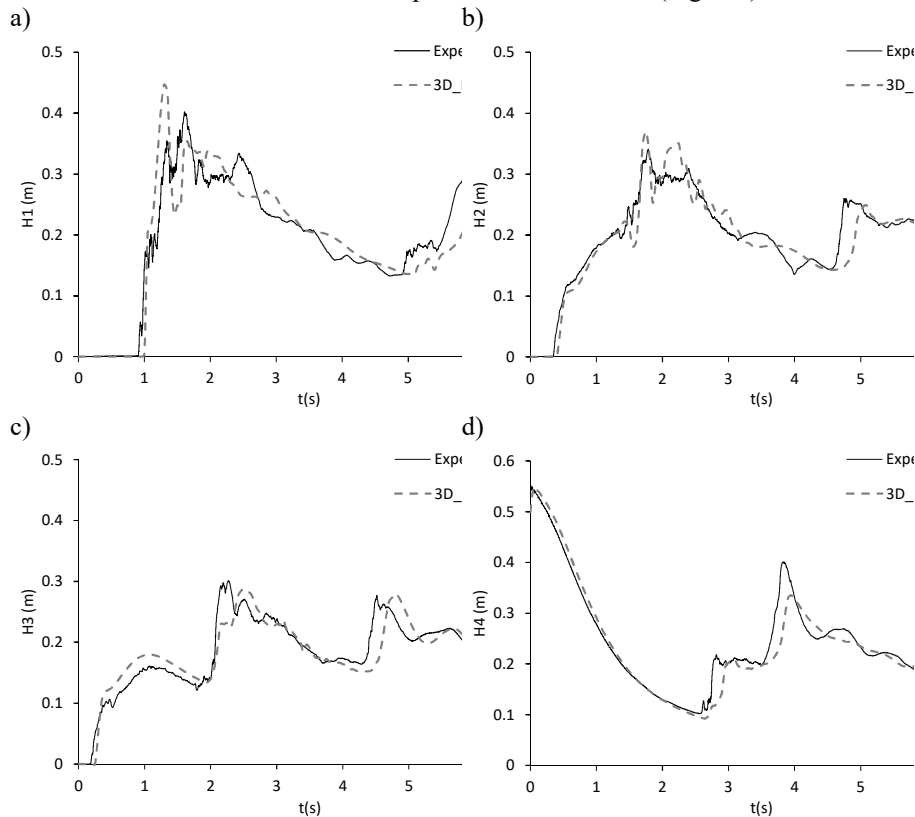


Fig. 3. Comparison of simulated and measured water depth in time
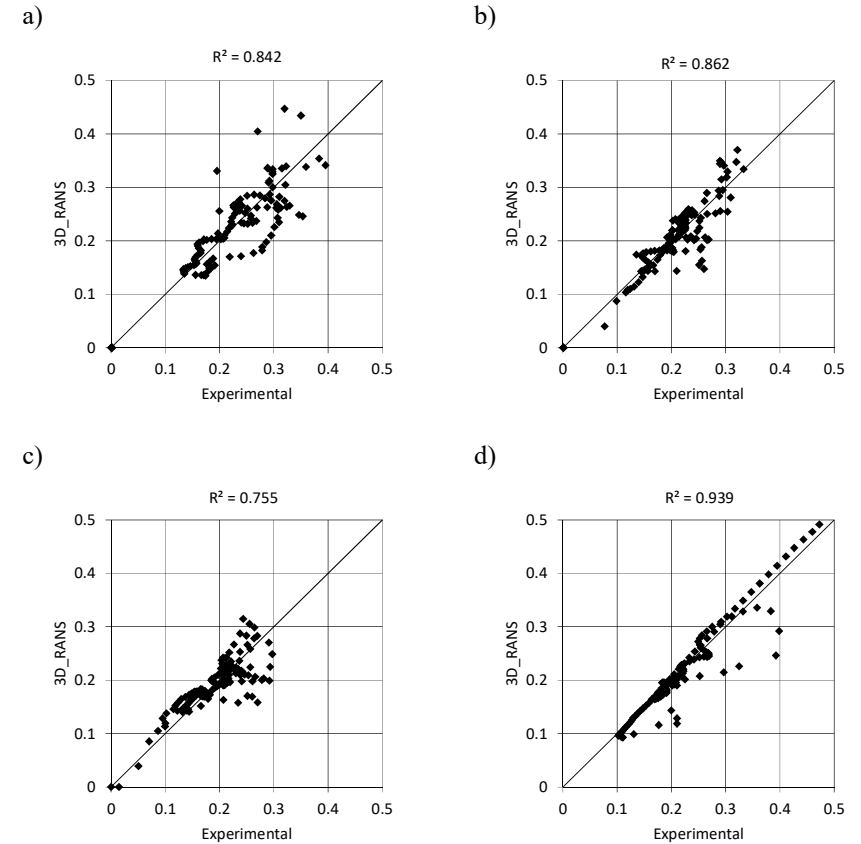a) H1, b) H2, c) H3, and d) H4



Fig. 4. Comparison of simulated and measured water depth (m)
a) H1, b) H2, c) H3, and d) H4

### 4.2 Pressure

Fig. 5 shows the variation in time of the pressure variable measured at different points of the obstacle. The points P1, P3, P5, and P7 are analyzed. A qualitative analysis shows a better capture of the temporal variation of the points in the frontal face of the obstacle (P1, P3) than those that are in the upper part of the obstacle (P5, P7). Numerical model results shows a time lag of peak value in the points P1 and P3. Quantitative analysis allows for qualitative assertion (Figure 6). Thereby, $R^2$ is 0.747 and 0.606 for pressure P1 and P3 respectively, whilst P5 and P7 are 0.595 and 0.576.
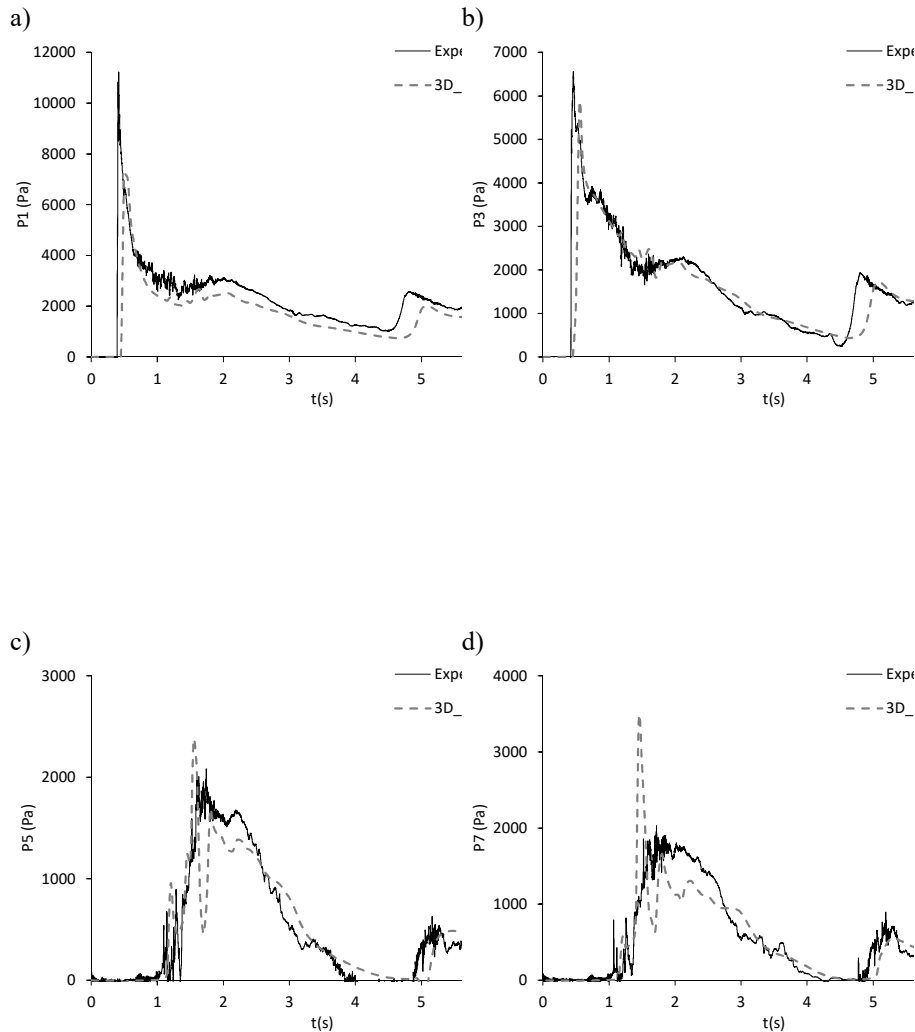
a)        b)

c)        d)

Fig. 5. Comparison of simulated and measured pressure in time
a) P1, b) P3, c) P5, and d) P7
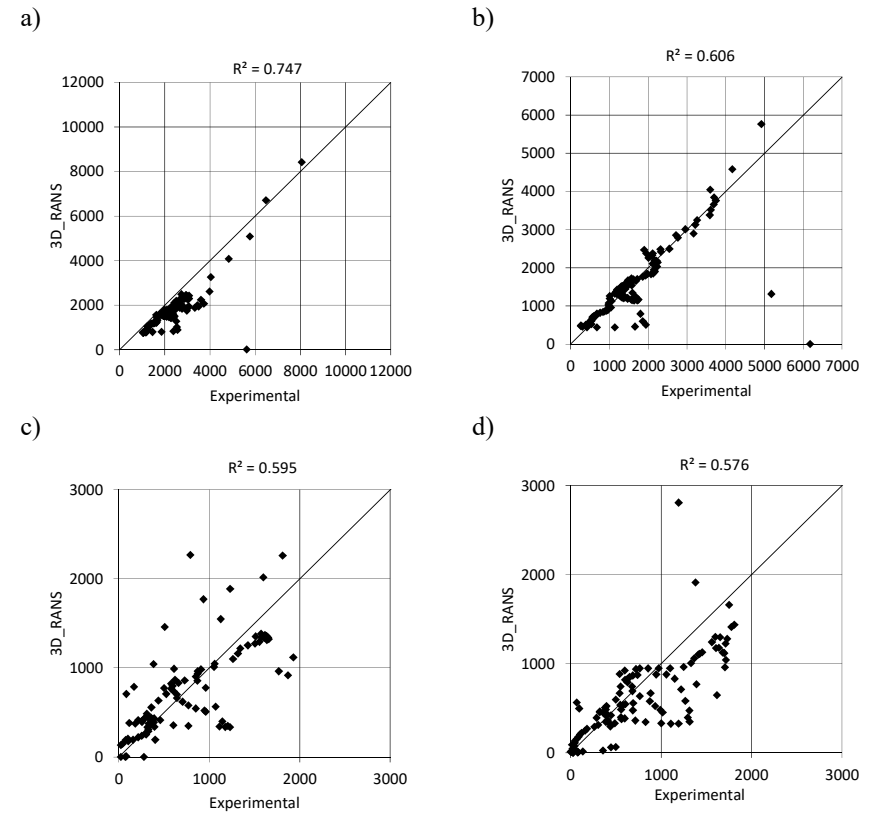
a)        b)

c)        d)

Fig. 6. Comparison of simulated and measured pressure (Pa)
a) P1, b) P3, c) P5, and d) P7

## 5. Conclusions

This study investigates the applicability of OpenFOAM code for the generation of flow field variables - water depth and pressure- in a dam-break laboratory scale using the RANS approach. The results demonstrate that the 3D numerical model configuration with RANS k-ε (RNG) approach can provide reliable results of the flow field in a dam-break case. Water depth values are reproduced better than pressure values by the 3D numerical model. Although the matching between the numerical solution and the physical experiment is quite promising, the application of the 3D numerical model for field-scale simulation would be computationally expensive.

## Acknowledgments

## References

[1]. R. F. Dressler, "Hydraulic Resistance Effect Upon the Dam-Break Functions*," J. Res. Natl. Bur. Stand. (1934)., vol. 49, no. 3, 1952.
[2]. L. Fraccarollo and E. F. Toro, "Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems," J. Hydraul. Res., vol. 33, no. 6, pp. 843–864, Nov. 1995.
[3]. S. Soares-Frazão and Y. Zech, "Experimental study of dam-break flow against an isolated obstacle," J. Hydraul. Res., vol. 45, no. sup1, pp. 27–36, Dec. 2007.
[4]. K. M. T. Kleefsman, G. Fekken, A. E. P. Veldman, B. Iwanowski, and B. Buchner, "A Volume-of-Fluid based simulation method for wave impact problems," J. Comput. Phys., vol. 206, no. 1, pp. 363–393, Jun. 2005.
[5]. X. Liu and M. H. García, "Three-Dimensional Numerical Model with Free Water Surface and Mesh Deformation for Local Sediment Scour," J. Waterw. Port, Coastal, Ocean Eng., vol. 134, no. 4, pp. 203–217, 2008.
[6]. A. S. Ramamurthy, S. S. Han, and P. M. Biron, "Three-Dimensional Simulation Parameters for 90° Open Channel Bend Flows," J. Comput. Civ. Eng., vol. 27, no. 3, pp. 282–291, 2013.

# Трехмерный численный анализ прорыва плотины с использованием OpenFOAM

[1, 2] *Эстебан Санчес-Кордеро <esteban.sanchezc@ucuenca.edu.ec>*
[2] *Мануэль Гомез <manuel.gomez@upc.edu>*
[2] *Эрнест Блейд <ernest.blade@upc.edu>*
[1] *Эквадор, г. Куэнка, Университет Куэнки,*
*Инженерный факультет, Департамент строительства*
[2] *Испания, Барселона, Политехнический университет Каталонии,*
*Научно-исследовательский институт FLUMEN,*
*Департамент строительства и охраны окружающей среды*

**Аннотация.** В статье представлен трехмерный численный анализ структуры поля течения при прорыве плотины в масштабах лаборатории путем. Численные вычисления основывались на методе конечных объемов (Finite Volume Method, FVM), OpenFOAM. В численной модели турбулентность обрабатывается с применением методологии RANS, а метод жидких объёмов (VOF, Volume of Fluid) используется для отслеживания свободной поверхности воды. Результаты вычислений оцениваются на основе экспериментальных данных. Для валидации численной модели используются измерения глубины и давления воды. Результаты показывают, что численные вычисления удовлетворительно воспроизводят изменения этих переменных во времени.

## Список литературы

[1]. R. F. Dressler, "Hydraulic Resistance Effect Upon the Dam-Break Functions*," J. Res. Natl. Bur. Stand. (1934)., vol. 49, no. 3, 1952.
[2]. L. Fraccarollo and E. F. Toro, "Experimental and numerical assessment of the shallow water model for two-dimensional dam-break type problems," J. Hydraul. Res., vol. 33, no. 6, pp. 843–864, Nov. 1995.
[3]. S. Soares-Frazão and Y. Zech, "Experimental study of dam-break flow against an isolated obstacle," J. Hydraul. Res., vol. 45, no. sup1, pp. 27–36, Dec. 2007.
[4]. K. M. T. Kleefsman, G. Fekken, A. E. P. Veldman, B. Iwanowski, and B. Buchner, "A Volume-of-Fluid based simulation method for wave impact problems," J. Comput. Phys., vol. 206, no. 1, pp. 363–393, Jun. 2005.
[5]. X. Liu and M. H. García, "Three-Dimensional Numerical Model with Free Water Surface and Mesh Deformation for Local Sediment Scour," J. Waterw. Port, Coastal, Ocean Eng., vol. 134, no. 4, pp. 203–217, 2008.
[6]. A. S. Ramamurthy, S. S. Han, and P. M. Biron, "Three-Dimensional Simulation Parameters for 90° Open Channel Bend Flows," J. Comput. Civ. Eng., vol. 27, no. 3, pp. 282–291, 2013.