ZIBELINE INTERNATIONAL PUBLISHING

ISSN: 1024-1752
CODEN : JERDFO

Journal of Mechanical Engineering Research and Developments (JMERD)
DOI : http://doi.org/10.26480/jmerd.04.2018.01.04

CrossMark

REVIEW ARTICLE

# EVALUATION OF VELOCITY VARIATION IN A CHUTE AS A FUNCTION OF ROUGHNESS

**Plua Frank[1*], Hidalgo Victor[2], Ortega Patricio[1], Ruiz Marcelo[1]**

*[1]Departamento de Ingeniería Civil y Ambiental, Escuela Politécnica Nacional, Quito, Ecuador.*
*[2]Departamento de Ingeniería Mecánica, Escuela Politécnica Nacional, Quito, Ecuador.*
**Corresponding Author Email: frank.plua@epn.edu.ec; frank.plua@gmail.com*



**ARTICLE DETAILS**

*Article History:*

Received 13 September 2018
Accepted 16 October 2018
Available online 7 November 2018

**ABSTRACT**

The present paper establishes numerical simulations with CFD packages for analyze the velocity variation as a function of roughness in a chute, and to determine the use of OpenFOAM® CFD package for this phenomenon. Experimental data was obtained from a physical hydraulic model with a smooth bottom and was compared with the results of numerical simulations from the CFD packages: ANSYS®, FLOW-3D® and OpenFOAM®. Subsequently, the roughness at the channel bottom was increased and new simulations were carried out for define the new velocity values. Increases in the resistance coefficient resulted in a significant reduction in velocity and the results of free open source OpenFOAM® are similar to those obtained in physical modeling as well as commercial software.

**KEYWORDS**

Chute, Roughness, Numerical Simulation, OpenFOAM®, ANSYS®, FLOW-3D®.

## 1. INTRODUCTION

A chute is a channel having steep slopes to discharges the flow from higher to lower elevation with free surface to high velocities. Chutes and spillways are designed to spill large water discharges over a hydraulic structure without major damage to the structure itself and to its environment [1]. Chute is usually used where the drop is greater than 4.5 m and where the water is conveyed over long distances and along grades that may be flatter than those for drops. This structure is essential to handle flows in a safe way and concentrate the dissipation of energy in a point as well as control it efficiently. The flow conditions in the chute will depend on physical characteristics such as geometry, roughness, material slope; and fluid characteristics such as flow, velocity, viscosity and temperature. The dimensions of the chute will depends on topography of the site and the conditions of project and its cross section is commonly rectangular.

Chutes are widely studied through physic and numeric modeling to predict how would operate under various flow conditions [1]. For steep chutes, both the flow acceleration and boundary layer development affect the flow properties significantly. The complete flow calculations can be tedious and most backwater calculations are not suitable because backwater calculations are valid only for fully developed flows. Furthermore, most softwares assume hydrostatic pressures distributions and neglect the effects of free surface aeration [2]. In chutes, processes of air entrainment and turbulence in the downstream bowl dissipate kinetic energy, resulting in a hydraulic jump [3].

The change of roughness of the channel´s material has been studied and modeled, displaying favorable results in the raising the resistance coefficient to dissipate energy [4]. Water flowing over a rough or steeped bottom is highly turbulent and can dissipate a major proportion of its energy [5]. Consequently, it is suitable to evaluate the variation of the velocity, by changing the roughness in the bottom through numerical modelling.

Computational Fluid Dynamics (CFD) has become a very important tool to evaluate hydraulic phenomena, particularly turbulence in Hydraulic Engineering [6]. In hydraulics, turbulence is the chaotic motion of particles in water, and it represents a basic problem in CFD methodologies [7,8]. ANSYS®-CFX and FLOW-3D® are examples of commercial CFD codes, while OpenFOAM® is one open source option for numerical modelling because it performed well in turbulence modeling for environmental fluxes [9]. ANSYS®-CFX is considered as one of the most popular codes, due to its facility in meshing constructions and FLOW-3D® has shown positive results in modeling CHUTES and in analyzing the dissipation of kinetic energy [10,11]. Finally, OpenFOAM® has also gained attention for the large number of turbulence models it has produced [12].

This works aims to determine by numerical simulation with CFD packages: ANSYS®-CFX, FLOW-3D® and OpenFOAM®, the velocities values produced in a chute when the roughness is modified, compare the results of the simulations and establish the reliability of free OPENFOAM package to investigate this type of phenomena.

## 2. METHODS

### 2.1 Experimental Method

The present investigation corresponds to a case study of the sewage system of the city of Quito in which the Water Supply and Sanitation Company of Quito (EPMAPS) hired Escuela Politécnica Nacional (EPN). The prototype is part of the derivation and diversion works of “El Batán” creek in Quito, Ecuador. These works convey the combined sewer from “Iñaquito” collector to the “El Batán” Creek. During periods of heavy rainfall, the flow causes the deterioration of the banks, and damage and sliding of the slopes, due to its erosive action.

The prototype is a chute that discharges approximately 170 $m^3/s$ in rainy season periods. The main channel of the structure has a horizontal length (L) of 67.78 m, slope of the bottom (Io) is 100%, and a width (w) of 10m

[13]. EPMAPS contracted EPN for the construction of two Hydraulic Models to analyze the phenomenon.

EPN executed investigation projects and engineer thesis on the Chute Hydraulic Model, scale 1:25 , that analyzes the hydraulic behavior, conveyance, dissipation of energy and final dispose of the water (Figure 1).


**Figure 1:** a) Chute Hydraulic Model 1:25 “El Batán”Creek, b) Chute Hydraulic Model 1:25 “El Batán”Creek

Experiments were carried out to obtain the depth and velocity in the chute with a smooth bottom for different ranges of flows according to the purposes of the studies executed. The following table shows the data obtained from the “Hydraulic Model Study of the derivation and diversion works of the El Batán Creek” and “Analysis of unstable and self-aerated flow in steep-slope channels” [13,14].

**Table 1:** Experimental data from Hydraulic Model of the Chute with smooth surface

| Flow (l/s)* | Section of chute | Velocity (m/s) | Study |
|---|---|---|---|
| 16 | Inlet | 5,13 | Escuela Politécnica Nacional,2000 |
| | Outlet | 4,38 | |
| 18,92 | Inlet | 4,62 | Haro,2006 |
| | Outlet | 3,98 | |
| 32 | Inlet | 4,32 | Escuela Politécnica Nacional,2000 |
| | Outlet | 4,57 | |
| 48 | Inlet | 4,60 | Escuela Politécnica Nacional,2000 |
| | Outlet | 6,22 | |

*1l/s = 1*10-3 m³/s

### 2.2 Numerical Simulation

After modeling the 1:25 scale chutes, the phenomenon was analyzed through numerical simulation. The phenomena was simulated by Computational Fluid Dynamics (CFD) methodology. Two of the CFD packages used were commercial products-- ANSYS® CFX and FLOW-3D®—while the other one was free and open source-- OpenFOAM®. These three CFD packages solved the RANS equations, using finite – volume and finite – differences methods.

Out of the two commercial products, ANSYS® develops and markets finite element analysis software used to simulate engineering problems, while FLOW-3D® is a CFD that analyzes various physical flow processes. Lastly, OpenFOAM® is an open-source software for computational dynamics (CFD), a toolbox for the development of numerical solvers, and pre-/post-processing utilities for the solution of continuum mechanics problems.

In regard to turbulence, there was used two different models used to make the comparative analysis. FLOW-3D® and ANSYS® CFX model used RNG K-ε; meanwhile, OpenFOAM® used the k-ω Shear Stress Transport (SST). The RNG k-ε base the equation in the eddy viscosity, $\nu_t = C_\mu \frac{k^2}{\epsilon}$, where $C_\mu$ is a proportion coefficient, $k$ is the kinetic energy of turbulence, and $\epsilon$ is the dissipation rate, such as in the standard model. However, in the RNG k-ε model, the coefficients assume different values. In addition, this model introduces another source term,$R$, in the dissipation rate equation, which is given for the equation (1)

$$R = \frac{C_\mu \eta^3 (1-\eta/\eta_0)}{1+\beta\eta^3}\frac{\epsilon^2}{k} \tag{1}$$

Where, $\eta_0 = 4.8$, $\beta = 0.012$ and $\eta = \left[(v_{i,j}+v_{j,i})\left(\frac{1}{2}\left[v_{i,j}+v_{j,i}\right]\right)\right]^{1/2}\frac{k}{\epsilon}$.
On the other hand, the k-ω model replaces the dissipation rate term, $\epsilon$, by the specific dissipation rate term, $\omega = \frac{\epsilon}{(C_\mu k)}$, where $C_\mu = 0.090$ [8]. The advantage is that the model doesn’t have a dumping function like the k-ε. The k-ω SST includes a cross diffusion term, $CD_{k\omega}$, in the transport equation for the specific dissipation rate by equation 2 [15]. In this model, Bradsjaw’s assumption is to define turbulent shear stress, $\tau = \rho a_1 k$, where $a$ is the turbulence structural parameter, whose value is constant, at nearly 0.16 [16].

$$CD_{k\omega} = max\left(2\rho\sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}, 10^{-20}\right) \tag{2}$$

Flows were modeled for smooth surface with the CFD packages. The next table shows the testing plan of the numeric simulation:

**Table 2:** Testing plan of the numeric simulation

| Test Number | Flow [lts/s] | OpenFOAM® | FLOW–3D® | ANSYS® CFX |
|---|---|---|---|---|
| 1 | 16,00 | ✓ | ✓ | ✓ |
| 2 | 18,92 | ✓ | | |
| 3 | 32,00 | ✓ | | |
| 4 | 48,00 | ✓ | ✓ | ✓ |

The absolute roughness used was 0.0015mm on the smooth surface. The numerical models were validated with velocities in the inlet and outlet of the structure. Once the convergence of parameters in the models was verified, the mass conservation law was also confirmed.

In the OpenFOAM® package, the selected mesh type was non-structured, composed of hexahedrons and tetrahedrons domains. The meshing contained 156,105 elements, 30 of which were tetrahedrons. The modeling time was 50 seconds and the simulated flows were 16, 18.92, 32, and 48 l/s. Maximum simulation time was 5 hours and the minimum was 4 hours. The INTERFOAM solver was used to compute the interaction between two compressive fluids due to the air entrainment present in the chute.

In the ANSYS® CFX, the type of mesh was mixed, 194840 hexahedrons and 100 tetrahedrons. The modeling time was 60 seconds, and the simulated flows were 16 and 48 l/s. The simulation time was 3 hours and 48 minutes. In the FLOW-3D®, the selected mesh was structured, composed of 525,316 hexahedrons. The modeling time was 60 seconds and the simulation time was 3 hours and 8 minutes.

## 3. RESULTS

Next table shows the results of velocities produced in the chute for a smooth surface with the CFD packages, which are compared to the results obtained in the experimental model:

**Table 3:** Velocity values in the chute-smooth surface-

| Flow (l/s) | VELOCITIES – SMOOTH SURFACE (m/s) | | | | | | | |
|---|---|---|---|---|---|---|---|---|
| | INLET | | | | OUTLET | | | |
| | EXPERIMENTAL MODEL | OpenFOAM® | FLOW-3D® | ANSYS® CFX | EXPERIMENTAL MODEL | OpenFOAM® | FLOW3D® | ANSYS® CFX |
| 16 | 5.13 | 4,44 | 4,7 | 4,46 | 4.38 | 4,71 | 5.16 | 5,73 |
| 18,92 | 4.62 | 4,19 | | | 3.98 | 4.98 | | |
| 32 | 4.32 | 4,73 | | | 4.57 | 6,25 | | |
| 48 | 4.60 | 5,11 | 7,85 | 5,68 | 6.22 | 6,86 | 6.38 | 8,14 |

For a rough contour, the roughness value of the channel bottom was modified from 0.0015mm to 0.53mm in all simulations, while the rest of the conditions were not changed. The next table shows the results:

**Table 4:** Velocity values in the chute-rough surface-

| Flow(l/s) | VELOCITIES – ROUGH SURFACE (m/s) | | | | | |
|---|---|---|---|---|---|---|
| | INLET | | | OUTLET | | |
| | OpenFOAM® | FLOW-3D® | ANSYS® CFX | OpenFOAM® | FLOW-3D® | ANSYS® CFX |
| 16 | 3,54 | 3,78 | 4,03 | 3,92 | 4.09 | 4,57 |
| 18,92 | 3,67 | | | 4,08 | | |
| 32 | 4,26 | | | 4,62 | | |
| 48 | 4,72 | 5,12 | 5,35 | 5,38 | 5.99 | 6,81 |

Figure 2 shows the results of numerical simulations with CFD packages for a smooth bottom in the inlet of a chute. In the range of 15 l/s to 20 l/s, the values of all numerical simulations are similar; from 20 l/s to 50 l/s, ANSYS® and OpenFOAM® are close to the experiments. From 32 l/s OpenFOAM® is the closest to the experimental results.

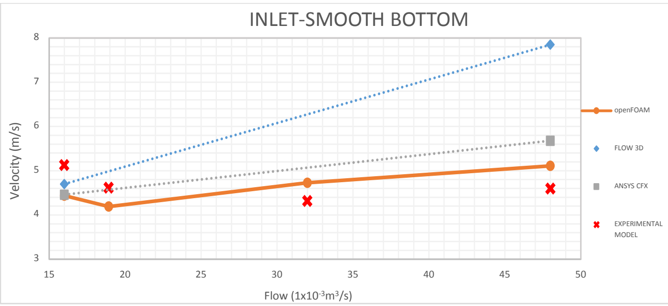


**Figure 2:** Results of all simulations for a smooth bottom in the inlet of a chute

Figure 3 presents the results of simulations for a smooth bottom in the outlet of a chute. For 16 l/s, OpenFOAM® is the closest to experimental value. From 18 a 50 l/s, the slope of ANSYS® simulation is similar to the slope of the hydraulic model data, and the results of the FLOW-3D® package are the closest to the experimental results.

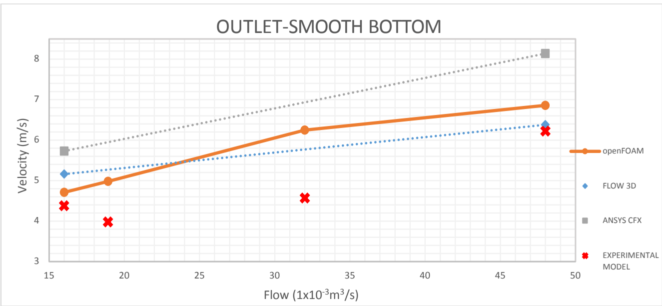


**Figure 3:** Results of all simulations for a smooth bottom in the outlet of a chute

Figures 4 and 5 shows the results of numerical simulations with the CFD packages for a rough bottom in the inlet and outlet of a chute. The values of all numerical simulations are similar as well as their slopes. OpenFOAM® presents the minimum values.

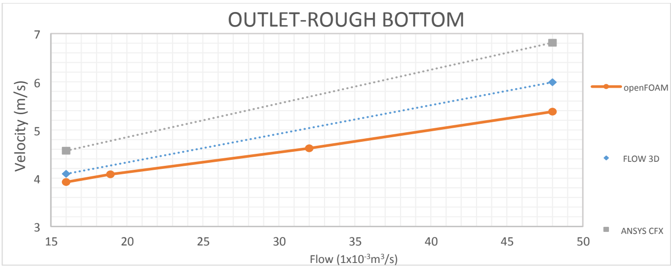


**Figure 4:** Results of all simulations for a rough bottom in the inlet of a chute

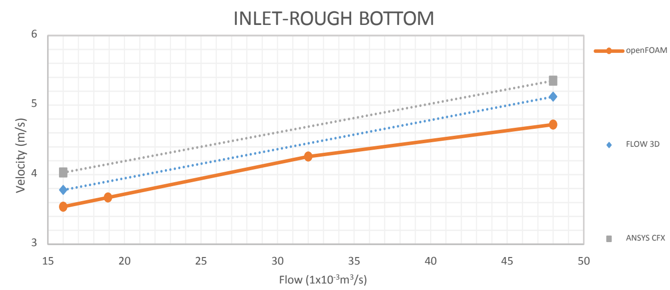


**Figure 5:** Results of all simulations for rough bottom in the outlet of a chute

Based on these results, the decrease in percentage velocity for the increase of the roughness was calculated. For 16 l/s, in the inlet of the chute OpenFOAM® and FLOW-3D® presents similar values with and average value of 20%. The calculated value with ANSYS® was 9.6%. For the outlet, velocity decreases in a range from 16.8% a 20.7%.

For 48 l/s, in the inlet, the percentage of reduction according to OpenFOAM® and ANSYS® goes from 5.8% to 7.6%. The value calculated with FLOW-3D® is 34.8%. For the outlet, the range of decrease velocity is 16.3% a 21.6 % (ANSYS® and OpenFOAM®, respectively). FLOW-3D® shows a value of 6.1%. See Table 5:

**Table 5:** Decrease of velocity for roughness increase

| Flow(l/s) | DECREASE OF VELOCITY FOR INCREASE IN ROUGHNESS (%) | | | | | |
|---|---|---|---|---|---|---|
| | INLET | | | OUTLET | | |
| | OpenFOAM® | FLOW-3D® | ANSYS® CFX | OpenFOAM® | FLOW-3D® | ANSYS® CFX |
| 16 | 20,3 | 19,6 | 9,6 | 16,8 | 20,7 | 20,2 |
| 48 | 7,6 | 34,8 | 5,8 | 21,6 | 6,1 | 16,3 |

## 4. DISCUSSION & CONCLUSIONS

The numerical modeling of the flow in the chute was carried out with CFD packages OpenFOAM®, ANSYS® and FLOW-3D® to determine the velocity values produced and compare the results of the simulations. It was determined that increases in the resistance coefficient resulted in a significant reduction in velocity. The free open source OpenFOAM® presented congruents results to those obtained in physical modeling as well as commercial software so its use is recommended to carry out research projects of this type of phenomena.

## REFERENCES

[1] Chanson, H. 2002. The Hydraulics of Stepped Chutes and Spillways, Tokio: A.A. Balkema Publishers.

[2] Baker, D.W., Reedy, P.E. 2008. World Environmental and Water Resources Congress, in Side-Channel Spillway Hydraulics (Case Study: Lake Skinner Spillway Adequacy Evaluation), Ahuapa'a.

[3] Chanson, H. 2002. "Hidráulica del flujo en canales abiertos", Bogotá: Mc Graw Hill.

[4] Khatsuria, R.M. 2005. Spillways and Energy Dissipators, United States of America: Marcel Dekker, ISBN: 0-8247-5789-0.

[5] Zafer, G., Aliihsan, K., Kenan, K. 2017. "Predicting the Numerical and Experimental Open-Channel Flow Resistance of Corrugated Steep Circular Drainage Pipes. Journal of Pipeline Systems Engineering and Practice, 8 (3), 0401-7004. DOI: 10.1061/(ASCE)PS.1949-1204.0000265.

[6] Gabl, R., Gems, B., Plörer, M., Klar, R., Gschnitzer, T., Achleitner, S., Aufleger, M. 2014. "Chaper 8: Numerical Simulations in Hydraulic," in Computational Engineering, New York Dorchret Londod, Springer International Publishing Switzerland, pp. 195-224.

[7] Emmanueli, G.S., Moritz, B. 2011. Hydromechanics: Theory and Fundamentals, Weinheim, Germany: WILEY-VCH Verlag GmbH & Co. KGaA.

[8] Wilcox, D.C. 2006. Turbulence Modeling for CFD (Third Edition), La Cañada, California: DCW Industries, Inc.

[9] Kasiteropoulou, D., Liakopoulos, A., Michalolias, N., Keramaris, E. 2016. Numerical Modelling and Analysis of Turbulent Flow in an Open Channel with Submerged Vegetation. Environmental Processes, 4 (17), 1-15, DOI: https://doi.org/10.1007/s40710-017-0235-x, 2016.

[10] Parsaie, A., Haghiabi, A.H., Moradinejad, A. 2015. CFD modeling of flow pattern in spillway's approach channel. Sustainable Water Resources Management, pp. 245-251.

[11] Tabari, M.M.R., Tavakoli, S. 2015. Effects of Stepped Spillway Geometry on Flow Pattern and Energy Dissipation. Arabian Journal for Science and Engineering, 41 (4), 1215–1224. DOI: 10.1007/s13369-015-1874-8, 2015.

[12] Liu, X. 2014. Chapter 2 Open-Channel Hydraulics: From Then to Now and Beyond, in Handbook of Environmental Engineering,15, New York, Springer Science+Business Media. DOI: 10.1007/978-1-62703-595-8_2, 128-158.

[13] Escuela Politécnica Nacional. 2000. Estudio en Modelo Hidráulico de las obras de derivación y desvío de la quebrada el batán, EPN, Quito, Ecuador.

[14] Haro, P.L., Jara, M.F. 2006. Análisis de FLujo inestable y auto aireado en canales de fuerte pendiente, Master Thesis, Quito, Ecuador: EPN.

[15] Menter, F.R., Kuntz, M., Langtr, R. 2003. Ten years of industrial experience with the SST turbulence model. Turbulence, heat and mass transfer, 4 (1), 625-632.

[16] Doussinault, M., Gleyzes, C., Aupoix, B., Gooden, J.H.M. 1998. Utilization of the Garteur swept wing data base for turbulence models' evaluation in boundary layer calculations. International Journal of Heat and Fluid FLow, 19, 431-438.

[17] Gemici, Z., Koca, A., Kaya, K. 2017. Predicting the Numerical and Experimental Open-Channel Flow Resistance of Corrugated Steep Circular Drainage Pipes. Journal of Pipeline Systems Engineering and Practice, 8 (3), pp. Art No. 04017004, DOI: 10.1061/(ASCE)PS.1949-1204.0000265.

[18] Fernández, J.M. 2012. Técnicas numéricas en ingeniería de fluidos: introducción a la dinámica de fluidos computacional (CFD) por el método de volúmenes finitos, Barcelona, España: Editorial Reverté.

[19] Poveda, R. 2005. Optimización de las estructuras de disipación de energía en pozos de bandejas, Master Thesis, EPN, Quito, Ecuador.

[20] Naudascher, E. 2000. Hidráulica de Canales abiertos, México DF, México: Limusa-Noriega.