



# IJRASET

International Journal For Research in  
Applied Science and Engineering Technology



---

# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

---

**Volume: 8      Issue: IX      Month of publication: September 2020**

**DOI: <https://doi.org/10.22214/ijraset.2020.31712>**

**[www.ijraset.com](http://www.ijraset.com)**

**Call:  08813907089**

**E-mail ID: [ijraset@gmail.com](mailto:ijraset@gmail.com)**

# CFD Modelling of the Dynamics of Flow in Open Flume and its Comparison with Experimental Results

Wasim Karam<sup>1</sup>, Imran Ullah<sup>2</sup>, Sawood<sup>3</sup>

<sup>1, 2, 3</sup>Department of Civil Engineering, University of Engineering and Technology, Peshawar

**Abstract:** *Computational Fluid Dynamics (CFD) provides a qualitative (and sometimes even quantitative) prediction of fluid flows. CFD enables scientists and engineers to perform ‘numerical experiments (i.e. computer simulations) in a ‘virtual flow laboratory’. Difference method.*

*UNET package. Fluid motion is controlled by the basic principles of conservation of mass, energy and momentum, which form the basis of fluid mechanics and hydraulic engineering. Complex flow situations must be solved using empirical approximations and numerical models, which are based on derivations of the basic principles (backwater equation, Navier-Stokes equation etc.). All numerical models are required to make some form of approximation to solve these principles, and consequently all have their limitations. The study of hydraulics and fluid mechanics is founded on the three basic principles of conservation of mass, energy and momentum. Real-life situations are frequently too complex to solve without the aid of numerical models. There is a tendency among some engineers to discard the basic principles taught at university and blindly assume that the results produced by the model are correct. Regardless of the complexity of models and despite the claims of their developers, all numerical models are required to make approximations. These may be related to geometric limitations, numerical simplification, or the use of empirical correlations. Some are obvious: one-dimensional models must average properties over the two remaining directions. It is the less obvious and poorly advertised approximations that pose the greatest threat to the novice user. Some of these, such as the inability of one-dimensional unsteady models to simulate supercritical flow can cause significant inaccuracy in the model predictions. In this research the flow in flow channel was modelled with the help of commercially available CFD software FLOW-3D with different discharges as boundary conditions and the depths at the boundaries are known the flow is simulated for different angles of the flow channel. The variation in flow parameters was modelled accordingly also the analytical solution of the solution was obtained and the phenomenon was modelled accordingly. The results shows close comparison between the results of the model and the experimental data. The results shows a good comparison between the model and the experimental data which indicates that FLOW-3D with RANS equation provide good approximation to flow characteristics prediction in open channels. Much work can be done by performing the statistical analysis of the data.*

**Keywords:** *Computational Fluid Dynamics, Navier Stoke Equation, FLOW-3D, Analytical Modelling. Open Flume*

## I. INTRODUCTION

Mathematics and especially numerical solutions play a very important part in the problems dealing with CFD. It is essential to make sure that the student is aware of the solutions of differential equations (including partial and non-linear differential equations) in analytical way. Of course, the student should also be aware of the limitations of obtaining solutions to these types of differential equations. For example, Navier-Stokes equations that are used widely throughout fluid dynamics are a good example for these types of non-solvable differential equations. However, this is where the power of numerical methods comes in as it allows you to use approximate solutions to these differential equations. There are numerous numerical methods that you can use for these types of solutions. However, with some differential equations (particularly those that deal with complex fluid flows), there are significant problems involved with their solutions even with numerical methods. Thus, this problem is dealt with by utilizing a computer to do the calculations for you in numerical methods. By increasing the number of iterations, your convergence on the correct solution increases greatly. Of course, it is also essential to state that none of the numerical methods even with the aid of supercomputers can produce exact solutions. You will have to use your own intuition to make sure that the necessary solutions are producing usable solutions. Also, there will be a certain percentage of error, depending upon the number of iterations that you are performing. In addition, your initial approximation of the boundary conditions as well as the grid mesh in which you define your problem will also be interest to you for more correct results.

Of course, no matter which method that you use, it is essential for you to make sure that you use the correct grid system for your analysis. The correct selection of the grid can make the difference between success and failure. Moreover, the proper selection of the grid will also ensure that there is less error when the end results are received.

Hromadka and Yen [4] reviewed the fundamental equations for modeling flow in topographic floodplains and channels. Through various simplifications and assumptions, they derived the diffusion formulation for modeling unsteady two-dimensional flow. Validation and verification of the DHM program is included in the report that includes testing for dam break scenarios vs. the USGS K-634 computer program. The report and software

Toombes and Chanson [1] also reviewed the basic equations which govern fluid motion and tested the performance of four popular hydraulic software packages for flow in a channel with a weir and hydraulic jump. The software programs they tested were HEC-RAS (steady mode), MIKE 11, MIKE 21, and FLOW-3D. Our current work supplements and expands upon their work, and as presented later, extends the results of their analysis by the inclusion of additional laboratory data and other computer modeling results. Many modelers have benchmarked their CFD codes

Ever since Kuipers and Vreugdenhill [16] developed the first two-dimensional finite difference model for solving the depth-averaged flow equations, many researchers have solved these equations or variations for predicting hydraulic jump characteristics in open channel flow regimes.[17,18] The advent of computational power and a better need to analyze hydraulic jump properties at a microscopic scale motivated researchers to solve the Navier–Stokes equations [19–21] for simulating hydraulic jump extent and location. Pineda et al. [22] solved the

hydraulic jump extent and location. Pineda et al. [22] solved the Navier–Stokes equations using computer program ANSYS CFX for predicting the jump characteristics. They noted that to arrive at good accuracy, special care with grid selection and entrance boundary conditions is crucial. Jowhar and Jihan [23] compared the performance of HEC-RAS and the 2-D Adaptive Hydraulics (ADH) software for predicting steady-state jump characteristics. Their results indicate that the jump location from ADH may be more accurate than those from HEC-RAS.

## II. METHODOLOGY

A flume of length 4.14 m was used for experimentation. The width of the flume is 0.08 m where the angles of the flumes that the experiment was performed are  $0^{\circ}, 1^{\circ}, 2^{\circ}, 3^{\circ}$ . The discharge was changed accordingly from 50 LPM, 100 LPM to 150 LPM. The depth was measured at every 50 cm, 200 cm and then 350 cm.

The 1, 2 and 3 of the 2<sup>nd</sup> row of the table represents the data taken at 50 cm, 200 cm, and 350 cm respectively. The numbers below these bold 1, 2 and 3 represents the depth of flow in channel.



Figure 1. Hydraulic Flume



Figure 2. Flow Channel

The Flow was also modelled in the commercially available CFD model FLOW-3D. The Flume was modelled in FLOW-3D with viscosity model being run. The variation in the depth of flow with the change in the slope of the channel was observed and then modelled in FLOW-3D. The model is then meshed and the mesh size is kept one fluid and single phase analysis was run. Three probe points are located at 50 cm, 200 cm and then 350 cm in the model.

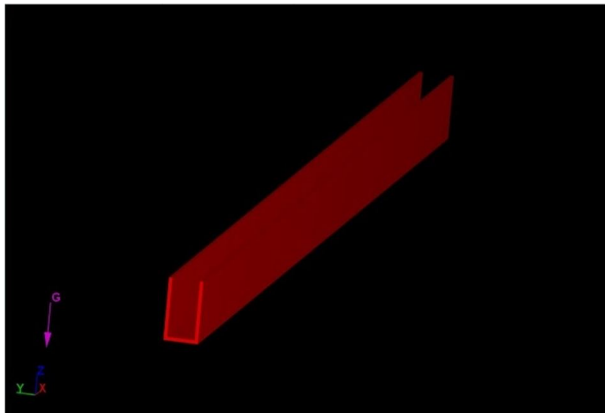


Figure 3. Modelled Channel in FLOW-3D

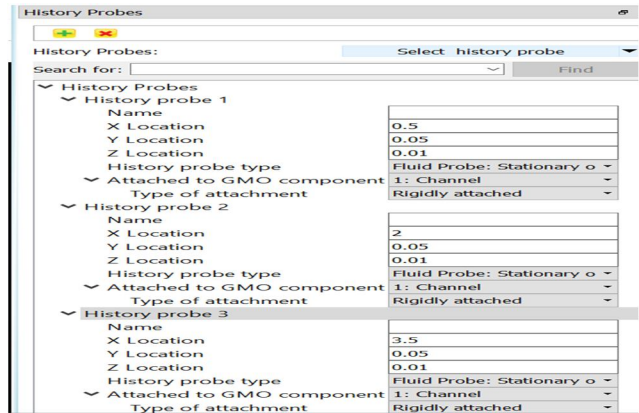


Figure 4. History Probe Details

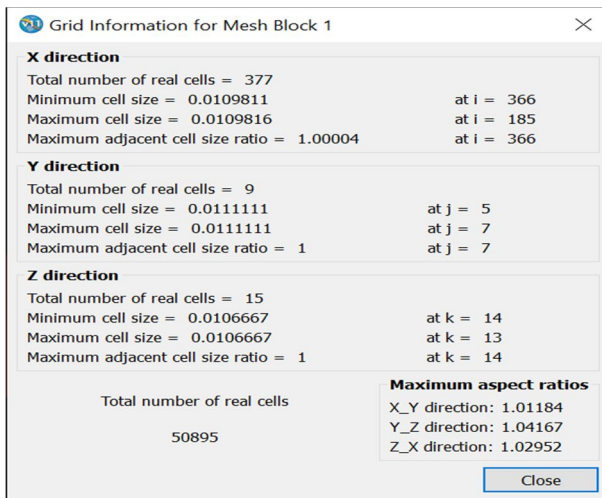


Figure 5. Mesh Information

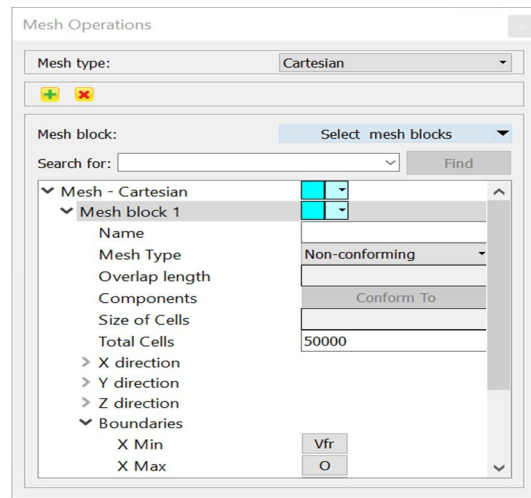


Figure 6. Mesh Block Operation

These three probe points are stuck to the .stl geometry file that is exported from Google Sketch Up with slopes of  $0^0$ ,  $1^0$ ,  $2^0$  and  $3^0$ . The mesh size is kept less than 0.01m for higher accuracy. The boundary conditions can be seen in the figure. With only the inflow rate at the X min while the other boundaries are kept accordingly. Each case was simulate for 50 sec making a total of 12 files each of 50 sec simulation.

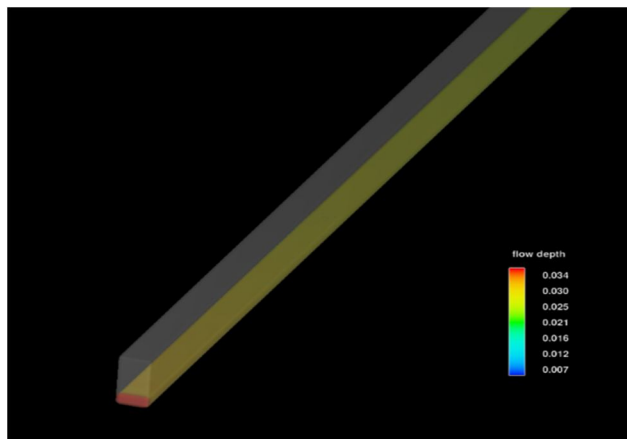


Figure 7. Analysis Window of FLOW-3D

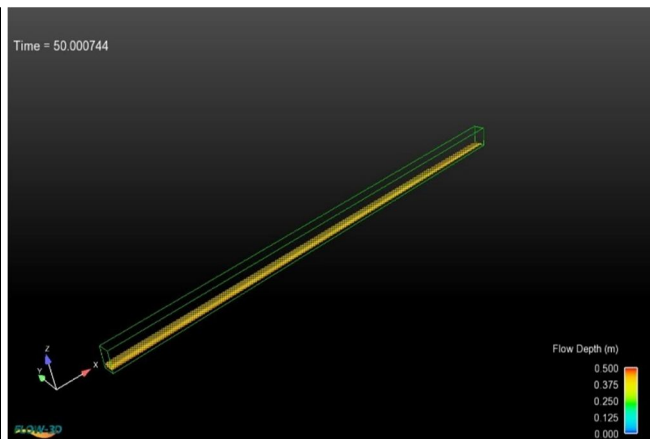


Figure 8. Analysis Window of FLOW-Sight

### III. RESULTS

The results that are obtained from the experimental procedure:

S. No	Angle	50 LPM				100 LPM				150 LPM			
		1	2	3	Avg.	1	2	3	Avg.	1	2	3	Avg.
1	0	4.5	4.7	4	4.40	5.6	6.2	5.8	5.87	8.1	9.5	9.8	9.13
2	1	1.8	1.8	2	1.87	2.2	2	2.4	2.20	2.7	2.5	2.7	2.63
3	2	1.7	1.5	1.8	1.67	1.8	1.8	2	1.87	2	1.8	1.7	1.83
4	3	1.3	1	1	1.10	1.6	1.3	1.2	1.37	2	1.8	1.5	1.77

Table 1. Experimental Data

The average of the flow depths are taken to know the average depth at the channel for the particular discharge are observed. From the graph below we know that the initial flow depth decreases immediately after the discharge is increased for the same angle. For a single discharge when the angle of the flow channel decreases the flow depth decreases consequently.

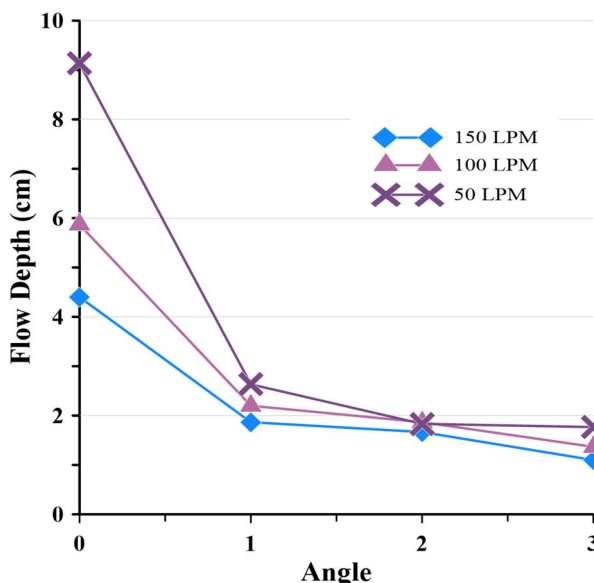


Figure 9. Discharge v/s Depth at Different Angles

Whereas from the CFD modelling the following results are obtained for the 3 probes that were set out:

S. No	Angle	FLOW-3D		
		50 LPM	100 LPM	150 LPM
1	0	4.69	6.1	9.3
2	1	2.01	2.23	2.75
3	2	1.60	1.50	1.6
4	3	1.02	1.20	1.5

Table 2. Numerical Data

The data shows close comparison between the depths readings that were taken for the different discharges that were taken in the experiment. The data shows that for lower angles i.e.  $0^{\circ}$  and  $1^{\circ}$ , The values are overestimated while for the higher angles  $2^{\circ}$  and  $3^{\circ}$ , The values are under estimated . The similarity in the data for greater angles is much higher as compared to the similarity in the data for lower angles.

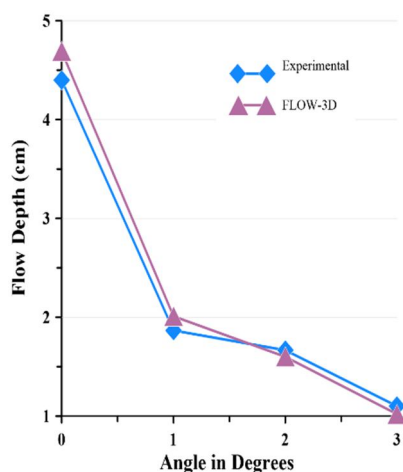


Figure 10. 50 LPM Comparison of Experimental and Numerical Data

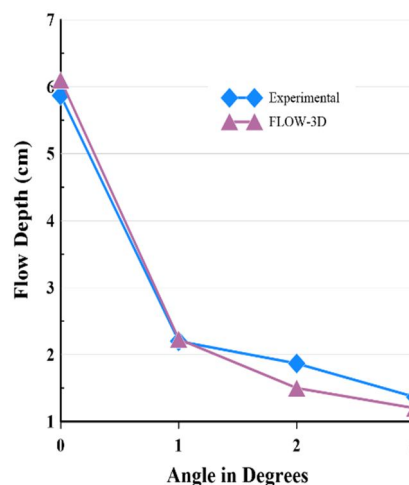


Figure 11. 100 LPM Comparison of Experimental and Numerical Data

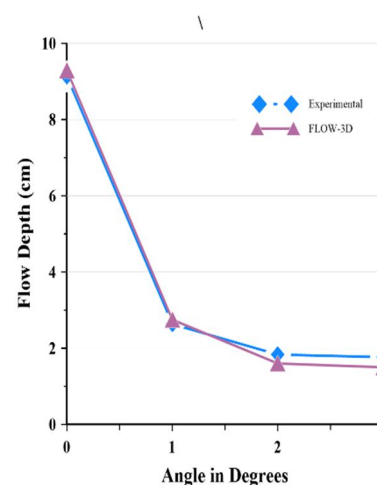


Figure 12. 150 LPM Comparison of Experimental and Numerical Data

#### IV. CONCLUSIONS

The data shows a good agreement between the experimental data and the numerical solution by FLOW-3D. There is pretty much similarity in the data for greater angles than for lower angles. The angles indicates the slope as the slope increases the depth decreases and the similarity for greater values is higher. The data shows that for horizontal slope there can be overestimation from the numerical model while for greater angles there is under estimation. The model works well for greater values rather than lower values of angles or slope.

#### REFERENCES

- [1] Gallegos, H.A.; Schubert, J.E.; Sanders, B.F. Two-dimensional high-resolution modeling of urban dam-break flooding: A case study of Baldwin Hills, California. *Adv. Water Resour.* 2009, 32, 1323–1335.
- [2] Kim, K.S. A Mesh-Free Particle Method for Simulation of Mobile-Bed Behavior Induced by Dam Break. *Appl. Sci.* 2018, 8, 1070.
- [3] Robb, D.M.; Vasquez, J.A. Numerical simulation of dam-break flows using depth-averaged hydrodynamic and three-dimensional CFD models. In *Proceedings of the Canadian Society for Civil Engineering Hydrotechnical Conference, Québec, QC, Canada, 21–24 July 2015.*
- [4] LaRocque, L.A.; Imran, J.; Chaudhry, M.H. 3D numerical simulation of partial breach dam-break flow using the LES and k-ε. *J. Hydraul. Res.* 2013, 51, 145–157.
- [5] Ritter, A. Die Fortpflanzung der Wasserwellen (The propagation of water waves). *Zeitschrift des Verein Deutscher Ingenieure* 1892, 36, 947–954.
- [6] Dressler, R.F. Hydraulic resistance effect upon the dam-break functions. *J. Res. Nat. Bur. Stand.* 1952, 49, 217–225.
- [7] Dressler, R.F. Comparison of theories and experiments for the hydraulic dam-break wave. *Int. Assoc. Sci. Hydrol.* 1954, 38, 319–328.
- [8] Whitham, G.B. The effects of hydraulic resistance in the dam-break problem. *Proc. R. Soc. Lond.* 1955, 227A, 399–407.
- [9] Stoker, J.J. *Water Waves: The Mathematical Theory with Applications*; Wiley and Sons: New York, NY, USA, 1957; ISBN 0-471-57034-6.
- [10] Marshall, G.; Méndez, R. Computational Aspects of the Random Choice Method for Shallow Water Equations. *J. Comput. Phys.* 1981, 39, 1–21.



10.22214/IJRASET



45.98



IMPACT FACTOR:  
7.129



IMPACT FACTOR:  
7.429



# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Call : 08813907089  (24\*7 Support on Whatsapp)