Numerical modeling of a liquid-gas interface flow in a channel

M Arévalo Silva¹, A Cárdenas Chinchilla¹, FJ Fossi², and J Garcia-Guarín¹,²

¹ Grupo de Investigación en Ingenierías Aplicadas para la Innovación, la Gestión y desarrollo, Universidad Francisco de Paula Santander, Ocaña, Colombia
² Grupo de Investigación en Desarrollo Tecnológico e Innovación, Servicio Nacional de Aprendizaje, Ocaña, Colombia

E-mail: manearevalo93@gmail.com, amcardenasc@ufpso.edu.co

Abstract. Water forms open channels, whether natural or artificial, which are studied in phenomena as in a flood; in order to save the greatest number of important people; important variables are the velocity profile and the pressure. The estimation of these variables is proposed using numerical model which is solved for the separate phases of liquid and gas using the Navier Stokes equation and for the liquid-gas interface the volume of fluid method is proposed, with two elements are estimated in a global way (liquid-gas) as if there were only one component. The equations are solved using the finite volume method. In addition, a mesh analysis is performed according to the channel geometry; the mesh types are hexahedral and tetrahedral with a number of elements respectively of 16791 and 166500. The tests carried out show a wave phenomenon in which the velocity of the flow increases from 0.7 m/s to 2.12 m/s and from 0.05 m to 0.07 m in height. In addition, an oscillatory movement was discovered which generated pressure gradients that varied between 0.2 kPa to 0.5 kPa. Tests are also carried out with other types of channels in which dry zones are observed, which are redesigned to reduce costs.

1. Introduction

This article was made in order to perform a numerical modeling of a liquid-gas interface flow in an open channel in which tests were performed through a laboratory guide available at the Francisco de Paula Santander Ocaña University where various phenomena, this article is divided into different practices, one of them the study with semi-empirical equations which limit the student with the mathematical part since these calculations are averages, trying to improve, evolve and compare with the method to propose volume of fluid used for multiple phases in order that the student can appreciate the units of the real physical phenomenon.

To achieve this research, laminar flow conditions were generated with the purpose of determining the field of velocities and pressures in an open channel. Then the necessary code was implemented in OpenFOAM to estimate the field of velocities and pressures in the open channel. And finally, the calculations were obtained by means of the equations that the software uses to determine the values of velocities and pressures and the change of its interface in the open channel; to achieve the numerical modeling of a liquid-gas interface flow in an open channel, the real channel was considered [1,2].

2. Materials and methods

The idea of modeling and analyzing the behavior of a laminar flow when it is conducted along a channel in which variations in velocity, pressure and liquid-gas interface can occur, with the intention
of contributing and improving academic aspects of the students of the Universidad Francisco de Paula Santander Ocaña who will carry out the practice [2].

Initially to achieve work in the case was made a selection of the place that has the required facilities, therefore the open channel is assumed that is available in the laboratory of hydraulics of the University Francisco de Paula Santander Ocaña, in order to obtain a data analysis before the behavior of the variables (velocities and pressures) and the channel, where these values are compared with those generated in the simulation using three free software.

The three free software’s in the programming were coupled to each other simultaneously. In the Salome software, the channel geometry was generated with the necessary meshing conditions and two types of meshes were selected to simulate, in the Openfoam software the mesh was verified, the border conditions and the initial conditions corresponding to the case were adapted of laminar incompressible flow to simulate and execute the code of the solver required for this case was executed (interfoam) and finally the software of Paraview which allows us to visualize the simulation and the values obtained, also gives us the option of exporting a table to Excel with the data found and there proceeds to analyze the data in the selected points and compare them with the points in practice [2].

The Openfoam software is a new and useful tool, in which several cases can be simulated in the open channel and an analysis of the equations that the software uses are made. By implementing the Navier-Stokes equations and the VOF fluid volume method, the results are compared with the semi-empirical method corresponding to the Manning and Chezy equations that are currently used, so that when applied in the everyday life can give a better solution to the problem [1, 3-7].

The modeling seeks to generate support at the time of the realization of functional prototypes in order to observe in a virtual way the purpose of a project, the benefit it provides is that it allows to appreciate the possible failures that can occur before its construction, this gives a financial savings.

3. Results and discussion

Below the results is presented to model the open channel.

3.1. Analysis of channel geometry

Channel of variable slope (CVS.): channel built in transparent acrylic of 8 mm of thickness that is protected by a metallic structure. This channel is attached through its metallic structure to protect the buffer tank. It is through CVS. That observed the hydraulic phenomena given in open channels. In addition, this channel can change its slope from (0 ° to 10 °) through a tilting mechanism. The channel has a safety brake or stop that serves as a guide to establish the 0 ° slope of the CVS [4].

In Figure 1 is possible to observe the distance of the points to be studied which refer to the location of 3 piezometers along the channel. Knowing the geometry of the available channel begins to implement the computation in this case for the design is used Salome software where it is dimensioned and given boundary conditions to the channel and processed to make a selection of the mesh types.

Figure 2 shows the location of all points of the geometry corresponding to the open channel in Cartesian coordinates of the x, y and z axes in the Salome software.

Table 1 shows the values of each of the points reflected in Figure 1, with respect to the axes x, y and z.
3.2. Type of mesh

For the study of the numerical modeling of a liquid-gas interface flow in a channel, two types of meshes are selected, which correspond to hexahedral and tetrahedral. In the following images a brief visualization is presented in the Salome software of what they are. The software of Paraview in each of them calculates two types of meshing and the behavior of the flow. In Figure 3 and Figure 4 are possible to appreciate the meshing of the geometry of the open channel of tetrahedron type, which has a value of 7800 elements and the geometry of the open channel of the Hexahedros type, which has a value of 54000 elements. Both made using the software of Salome. Above Figure 3 and Figure 4 the flow profile of the free channel is shown.

Table 1. Coordinates according to each point.

| Points | X (m) | Y (m) | Z (m) | Points 2nd part | X (m) 2nd part | Y (m) 2nd part | Z (m) 2nd part |
|--------|-------|-------|-------|-----------------|---------------|---------------|---------------|
| 1      | 0     | 0.00  | 0.015 | 7               | 6.98          | 0.00          | 0.00          |
| 2      | 0     | 0.32  | 0.015 | 8               | 6.98          | 0.32          | 0.00          |
| 3      | 0     | 0.32  | 0.146 | 9               | 6.98          | 0.32          | 0.13          |
| 4      | 0     | 0.32  | 0.495 | 10              | 6.98          | 0.32          | 0.48          |
| 5      | 0     | 0.00  | 0.495 | 11              | 6.98          | 0.00          | 0.48          |
| 6      | 0     | 0.00  | 0.146 | 12              | 6.98          | 0.00          | 0.13          |

It is observed that the movement of the flow is ordered with the Reynolds number less than 2000, which indicates that a laminar flow was worked and the values corresponding to the velocity and pressures at a selected point in the Hexaedros mesh [5].

For the modeling, Openfoam software is selected, which is new and has the necessary interFoam code to be implemented in the case of multiple phases (liquid-gas). Start conditions were assigned for speed and pressure, water at each border and for gravity, transport and flow properties were granted, the mesh was conditioned to be static and a definition of a time interval was made [4].

In Figure 5 and Figure 6 for the tetrahedral and hexahedral mesh are possible to appreciate, through the software of Paraview, the initial and final behavior of the pressures and the fluid inside the open channel, differentiated by the colors yellow, red and blue.
3.3. Governing equations used in modelling

3.3.1 Continuity equation. In Equation (1) is shown the velocity gradient [8].

\[ \nabla \cdot \mathbf{U} = 0 \]  

(1)

3.3.2 Equation of momentum. In Equation (2) is shown that represents the sum of forces of the fluid.

\[ \frac{\partial \mathbf{U}}{\partial t} + \nabla (\mathbf{U} \cdot \mathbf{U}) - \nabla \cdot (\gamma \nabla \mathbf{U}) = - \nabla P \]  

(2)

Where \( \frac{\partial \mathbf{U}}{\partial t} \) → Temporary derivative, \( \frac{\partial \mathbf{U}}{\partial t} = a \), \( \nabla (\mathbf{U} \cdot \mathbf{U}) \) → convective term, is a proper term of the fluids which represents the movement of the particle, \( \nabla \cdot (\gamma \nabla \mathbf{U}) \) → diffusive term takes care of the properties of the particles and \( \nabla P \) → source term [9].

3.4. Navier-Stokes equation

In Equation (3) is shown the Navier-Stokes Equation.

\[ \frac{\partial \mathbf{U}_i}{\partial t} + \mathbf{U}_i \frac{\partial \mathbf{U}_i}{\partial x_i} = - \frac{\partial P}{\partial x_i} + \gamma \frac{\partial^2 \mathbf{U}_i}{\partial x_i^2} + S_i + \rho g k \]  

(3)

\( \frac{\partial \mathbf{U}_i}{\partial t} \) → Transitory term, \( \mathbf{U}_i \frac{\partial \mathbf{U}_i}{\partial x_i} \) → advective term, \( \frac{\partial P}{\partial x_i} \) → change of pressure, \( \gamma \frac{\partial^2 \mathbf{U}_i}{\partial x_i^2} \) → Matrix, \( S_i \) → Losses and \( \rho g k \) → Effect of gravity on the fluid [8, 10].

3.5. Volume of fluid method (VOF)

In Equation (4) is shown the interface equation.

\[ \frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \mathbf{U}) = 0 \]  

(4)

\( \frac{\partial \gamma}{\partial t} \) → Derive local and \( \nabla \cdot (\gamma \mathbf{U}) \) → advective term [9].

The software has a condition in the programming which is in charge of recognizing the process that is going to work since there are two ways. When working with a single fluid, either liquid or gas, the software recognizes this condition and executes the continuity and Navier-Stokes equations. When working with multiphase flow, the software recognizes this condition and adds the equation of the VOF method to the solver, which corresponds to multi-phase cases (liquid-gas) [1].

3.6. Pressure

The Equation (5) for the pressure in the solver is as follows [2,11]:

\[ P_{\text{rg}} = p - \rho gz \]  

(5)

The piezometric tube is open to the atmosphere, which is subjected to a height analysis taking into account the condition of stagnation. The fluid height is determined using the Bernoulli Equation (6) and the input data is corrected for use in the Openfoam software [12,13].

\[ \frac{p_1}{\gamma} + \frac{V_1^2}{2g} + h_1 = \frac{p_2}{\gamma} + \frac{V_2^2}{2g} + h_2 \]  

(6)
With the data of position 1 and 2 the equation is simplified and $h_1 > h_2$ then $y_n = h_1 = \frac{V_2^2}{2g} + h_2$.

Table 2 shows the comparison of speed values for 3, 4 and 5 turns where $V_2 = \sqrt{V_x^2 + V_y^2 + V_z^2}$, $h_2 = z_2 = z + \text{Height of the slope}$, $z_2 = z + 0.015$.

In Table 2 is depicted the comparison between Manning, Chezzy and Tetrahedral Open Foam [14,15].

**Table 2. Comparison of speed data.**

| Method               | Piezometer 3 turns | Piezometer 4 turns | Piezometer 5 turns |
|----------------------|--------------------|--------------------|--------------------|
|                      | 16     | 30   | 44   | 16     | 30   | 44   | 16     | 30   | 44   |
| Manning [3,6,13]     | 0.64   | 0.64 | 0.59 | 0.68   | 0.68 | 0.64 | 0.68   | 0.68 | 0.64 |
| Chezzy (b) [14,15]  | 0.33   | 0.33 | 0.28 | 0.37   | 0.36 | 0.33 | 0.37   | 0.36 | 0.33 |
| Chezzy (k) [14,15]  | 0.68   | 0.68 | 0.62 | 0.74   | 0.72 | 0.68 | 0.74   | 0.72 | 0.68 |
| Openfoam (T)        | 0.68   | 0.68 | 0.62 | 0.68   | 0.62 | 0.68 | 0.62   | 0.68 | 0.62 |

Table 3 shows the comparison of data on the values of the velocities found by means of semi-empirical methods or approximate methods used in the practice of hydraulics laboratory of the Francisco de Paula Santander Ocaña University, with respect to the results obtained by means of the governing equations generated by the Interfoam solver.

**Table 3. Data comparison of the tetrahedron meshes speed.**

|                  | Real | Openfoam | Real | Openfoam | Real | Openfoam | Real | Openfoam |
|------------------|------|----------|------|----------|------|----------|------|----------|
|                  | 0.587| 0.938    | 0.691| 1.034    | 0.704| 0.710    | 0.704| 1.218    | 0.587| 1.044    |
|                  | 0.587| 1.211    | 0.691| 0.734    | 0.691| 1.204    | 0.704| 1.072    |

In Table 4, which represents the total pressure and hydrostatic pressure, the comparison of data of the heights found in the laboratory practice of hydraulics of the Francisco de Paula Santander Ocaña University is observed, which does not take into account the condition of stagnation, with respect to the results obtained for the height determined by the software plus the slope of the channel. The values obtained in the hexahedron mesh are observed for the pressure generated by the Interfoam solver and the hydrostatic pressure within an open channel.

**Table 4. Pressure scalar and gravitational (rgh).**

| Piezometer | 3    | 4    | 5    | 3    | 4    | 5    | 3    | 4    | 5    |
|------------|------|------|------|------|------|------|------|------|------|
| 16         | 336.1| 330.2| 336.1| 330.2| 324.2| 24.222| 18.329| 17.763| 414.7| 485.20| 487.64|
| 30         | 309.1| 309.8| 271.9| 309.1| 309.8| 271.9| 14.345| 15.257| 14.497| 333.8| 370.98| 385.86|
| 44         | 205.9| 381.6| 322.1| 205.9| 381.6| 322.1| 15.694| 15.505| 15.252| 231.4| 277.43| 279.20|

**4. Conclusion**

Throughout the realization of this article, it is worth mentioning the difficulty in defining the correct multi-phase model due to the number of options among conditions, parameters and models. That is why there has been a great variety of cases to simulate which contribute to the beginning of the study of open channels. The case was validated by taking experimental results of flow in a real open channel, obtaining satisfactory results. After making the comparison with the results obtained for the Manning and Chezy equation, the tetrahedral mesh showed values closer to the hexahedral mesh. Therefore, its use is recommended for future work. Also, the field of pressures is visualized over time, taking gravitational effects. A brief comparison is made in three different piezometers (16, 30, 44)
located along the channel, it can be deduced that the highest pressure is occurring in the position of the piezometer 16 and the lowest in the piezometer of 30, despite the fact that two occasions this presents higher than that of the piezometers 44, a brief sample is presented for 3, 4 and 5 turns of the valve of passage of the fluid in the piezometer 16 it is seen that the pressure is 336.104 Pa, in the piezometer 30 is 309.1421 Pa while in piezometer 44 the pressure corresponds to 205.887 Pa. The mesh made with tetrahedral gives higher values for the initial pressures.

References
[1] Deshpande S, Anumolu L and Trujillo M 2012 Evaluating the performance of the two-phase flow solver interFoam. *Comput. Sci. Discovery* 5(1) 014016
[2] Shuard A, Mahmud H and King A 2016 *IOP Conference Series: Materials Science and Engineering* (Miri) (Bristol: IOP Publishing) 121 012018
[3] Rizalihadi M, Shaskia N and Asharly H 2018 *IOP Conference Series: Materials Science and Engineering* (Banda Aceh) (Bristol: IOP Publishing) 352 012039
[4] Chow VT 1959 *Open-channel hydraulics* (Nueva York: McGraw-Hill)
[5] González G, Prieto N and Mercado I 2018 Large Eddy Simulation (LES) aplicado a un lecho fluidizado gas-sólido. parte i: reactor a escala de laboratorio *Revista UIS Ingenierías* 17(1) 93-104
[6] Jumain M, Ibrahim Z, Ismail Z, Entalai K, Makhtar M, Rahman M and Alias N 2018 *IOP Conference Series: Journal of Physics: Conference Series* (Johor) (Bristol: IOP Publishing) 1049 012027
[7] Yusof K W, Muhammad M M, Mustafa M R U, Zakaria N A and Gahani A 2017 *IOP Conference Series: Materials Science and Engineering* 216 012046
[8] Versteeg H and Malalasekera W 2005 *Introduction to computational fluid dynamics* (Malaysia: Longman Scientific & Technical)
[9] Márquez S 2004 Description and utilization of interFoam multiphase solver *Anales de Ingeniería Mecánica* 13 1-64
[10] Duran-Pinzón M, Páez-Arango J and García-Guarín P 2018 Numerical modeling and experimental analysis for flows in a homogeneous porous medium through soils *Iteckne* 15(1) 24-33
[11] Mavares F and Pertuz A 2017 Cementación de revestidor en flujo de gas/agua presurizado superficial no esperado en un campo de desarrollo costa afuera: Rio de Janeiro, Brasil. *Revista UIS Ingenierías* 16(2) 79-92
[12] Çengel Y A, Cimba J M, Olguín V C and Skarina S F 2006 *Mecánica de fluidos: fundamentos y aplicaciones* (São Paulo: McGraw-Hill)
[13] Binz M Rohlf W and Kneer R 2014 Direct numerical simulations of a thin liquid film coating an axially oscillating cylindrical surface *Fluid Dynamics Research* 46(4) 041402
[14] Dyakonova T and Khoperskov A 2018 *IOP Conference Series: Journal of Physics: Conference Series* (Voronezh) (Bristol: IOP Publishing) 973 012032
[15] Mejía F 2004 El transporte de cantidad de movimiento en canales *Revista EIA* (2) 17-33