

## Flow study in channel with the use computational fluid dynamics (CFD)

W. D. Oliveira<sup>1</sup>, M. S. G. Pires<sup>1</sup>, L. M. Canno<sup>1</sup>, L. C. L. J. Ribeiro<sup>1</sup>

<sup>1</sup>Unicamp – State University of Campinas – College of Technology  
R. Paschoal Marmo 1888, Centro, CEP 13484-332, Jd. Nova Itália, Limeira, SP,  
Brazil.

E-mail: wesleydias96@gmail.com, marta@ft.unicamp.br, laura@ft.unicamp.br,  
lubienska.ft@gmail.com

**Abstract.** The Computational Fluid Dynamics (CFD) is a tool used to numerically simulate fluid flow behavior, and all the laws that govern the study of fluids is the mass transfer and energy, chemical reactions, hydraulic behaviors, among others applications. This tool mathematical equation solves the problem in a specific manner over a region of interest, with predetermined boundary conditions on this region. This work is to study the flow channel through the CFD technique.

### 1. Introduction

The flow is classified into free surface, or simply free if, whatever the transversal section, the liquid is always in contact with the atmosphere. This is the situation of the flow in rivers, streams or channels. Rain water galleries, tunnels, channels and, gutter work as free conduits collectors drains. As characteristics of this type of flow, it can be said that it necessarily occurs by gravity and that any disturbance in localized portions can lead to changes in the cross section of the chain in other parts [1]. Therefore, are considered channels all conduits leading water with a free surface, with open or closed section [2].

Considering a channel the movement is continuous it is necessary that no new entries or out of new liquid occurs, so the quantity of liquid remains constant thereby maintaining the flow in the channel. If the flow section will not change, and the nature of the walls remain the same, while maintaining the constant depth and speed, the motion is considered uniform and the channel will also be considered uniform.

The most common flow when the fluid is water, which has relatively low viscosity, are classified as turbulent. In the case of free turbulent flow, ie those that occur in channels usually be divided into fluvial and torrential regime. The river is characterized when the average speed in a particular section is less than a certain critical value. The torrential is characterized when the average speed in a particular section is greater than a certain critical value.

In fluid flow, the numerical model is established through the classical transport equations: conservation of momentum, mass and energy, combined with the turbulence models [3] [4] [5] [6]. To reproduce the flow, from a mathematical point of view, the resolution of the equations is highly complex, which leads to the need to use computers. A strongly recommended methodology to solve these models is the Computational Fluid Dynamics (CFD), which aims to numerically solve the equations to simulate fluid dynamics, and thus solve many practical problems.



The computational model should be able to describe the physical behavior, mimicking the experimental system behavior; should be evaluated and compared to experimental, as the end result numerical system; must be capable of supporting theories or hypotheses, which explain the observed behavior; must be able to predict future behavior, that is, the effects produced by changes in system variables or mode of operation [7].

Currently, there are several commercial software using the computational fluid dynamics, such as PHOENICS CFX, STAR-CD, FLUENT, FLOW-3D and ANSYS CFX. In a simplified manner, these tools describe complex surfaces that are subdivided into numerous particles that form the mesh, in each of these particles are carried algebraic equations that are related to specific parameters of the problem. The resulting set of equations will be solved interactively, resulting in a complete description of the flow over the entire area. Therefore, the purpose of this paper is to study channels using CFD.

## **2. Computational Fluid Dynamics (CFD)**

The Computational Fluid Dynamics (CFD) is characterized as numerical simulation of phenomena involving fluid flow. Applications of this technique of analysis and simulation of industrial processes started in 1995, a part of the process, equipment or an accessory, was examined individually in order to obtain information about the flow patterns that can support a performance increase study [8]. CFD technique can be used in: - aerodynamics and spacecraft vehicles; - Hydrofoil vessels; - Combustion engines and gas turbines; - Flow within the diffuser; - Cooling equipment, including micro-circuits; - Chemical process engineering: mixing and separation, polymer modeling; - Environmental engineering: distribution of pollutants and effluents; - Hydrology and oceanography: flows in rivers, estuaries and oceans; - Meteorology: weather; - Biomedical engineering: blood flow through arteries and veins; - Rotor pumps in order to optimize the geometry; - Heat exchangers, in order to maximize thermal exchanges for analysis of the effects of the use of baffles and vanes and alternative geometric configurations; - Cyclone of the catalytic cracking units oil in order to develop designs with high collection efficiency and low pressure drop; - Tanks reactors agitated as no ideal model reactor for the prediction of dead zones and circulation and the effects of baffles and stirrers in mixing conditions; among others [9].

The equations of the mathematical model used by CFD are subjected to initial and boundary conditions, and represent a particular problem. The analytical solution of these equations is possible only for very simple flows. To analyze real problems numerical methods are used. Traditional methods for the numerical solution of differential equations are the Finite Differences Methods, Finite Volumes and Finite Element. In commercial packages, preference is given to the Finite Volumes depending on their robustness and their conservative characteristics [10]. Through CFD simulations can predict data such as power consumption, flow pattern and solids concentration. It is therefore a powerful tool for predicting three-dimensional flow and distribution of pollutant concentration.

CFD technology has become a key part in the design and analysis of products and processes of many companies, due to their ability to predict the performance of equipment and processes even before they are produced or implemented. Due to the increased processing power of computers, the evolution of graphics and interactivity in the 3-D image manipulation tool has become less expensive, reduced the simulation time and hence the cost of use [11].

These models are useful in granting processes and frameworks of water bodies, produce estimates of the concentration of conservative and non-conservative pollutants or releases are continuous or instantaneous [12].

## **3. Results and Discussions**

The system studied is composed of a channel portion constructed in reduced scale, corresponding to approximately 9 meters long with section 20x20 cm and two large reservoirs buried who assisted the circulation process of the fluid (Figure 1). In all studies that use such tools (computational fluid dynamics) experimental results are necessary because serves as comparative confirmation of results

obtained through studies. We raised all the characteristics of the channel by sampling and following the pollutant plume (Figures 2 and 3). The channel with the collectors can be seen in Figure 4.



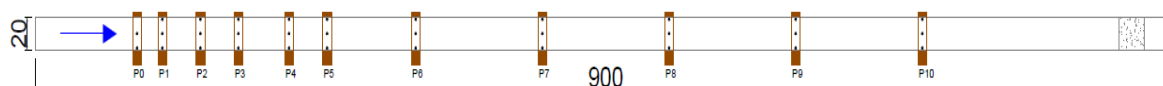
**Figure 1.** Channel stretch.



**Figure 2.** System Collector.



**Figure 3.** Channel with collectors.

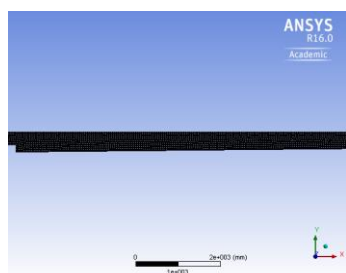


**Figure 4.** Profile channel with collection points.

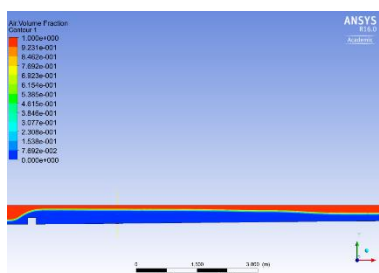
With the data collected from surveys carried out in the channel (Figure 5), it was possible to conduct studies in the CFD. The first stage of studies in the tool is the construction of the channel geometry and the study area mesh, called a pre-processing stage. The mesh refinement is directly related to the accuracy of the results presented by the simulation and, it is a very laborious step (Figure 6). Then comes the simulation, the first processing stage, which defines the domain and the boundary conditions of the study area, the fluid properties and initial conditions for convergence processing (Figure 7). The second stage of processing is the resolution of the transport equations. And last post-processing, where is possible view the results. The values found experimentally and simulated are similar (Figure 8). With the modeled system it is possible predict future behavior, both in case of changes in system variables or in operation mode.

Característica	Pontos										
	P0	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10
Área Molhada (m <sup>2</sup> )	0,029	0,031	0,032	0,033	0,034	0,035	0,037	0,04	0,043	0,046	0,049
Velocidade (m/s)	0,448	0,419	0,406	0,394	0,382	0,371	0,351	0,325	0,302	0,283	0,265
Vazão (m <sup>3</sup> /s)	0,013	0,013	0,013	0,013	0,013	0,013	0,013	0,013	0,013	0,013	0,013

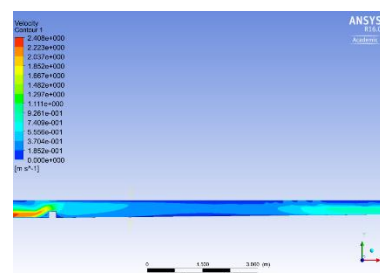
**Figure 5.** Characterization of channel collection points.



**Figure 6.** Mesh Definition.



**Figure 7.** Volume setting to be simulated.



**Figure 8.** Simulation speed in the channel.

#### 4. Conclusion

Even if the results provided by the computational technique are limited by the parameters used in the simulation, as in the experimental method, for the user is very easy to change parameters such as geometry, temperature and speed, as these are only "input data" for the simulator. In this sense, computational techniques are much closer to experimental than theoretical. So nowadays it is common to find the term "numerical experiments" in reference to the simulations of the same phenomena performed with different parameters [13]. The three strategies are compared to solve the fluid mechanics problems: experimental technique has more realistic advantages and present as disadvantages the required equipment, scale problems, simulation difficulty operating costs; Theory has the advantage of being more general, closed formula and as disadvantages restricted to simple geometries and physical processes generally restricted to linear problems; the numerical technique has the advantages the no restriction on linearity, geometries and complicated processes, temporal evolution of the process and disadvantages truncation errors, prescription of the appropriate boundary conditions, computational costs [14]. Despite all the flexibility that computational technique offers, it cannot solve many of the real problems involving flow fluid [15].

The channels through CFD studies are still scarce. Thus, it was proposed for this research some channel experimental studies in laboratory. It is possible to see that the model was able to describe the physical behavior, mimicking the experimental system behavior. CFD can be used to validate a component in the project stage and can be used to find ways to improve the characteristics of components already implemented. The simulation of computational fluid dynamics is practical and can change design feature with simple and fast modification in the Model geometry.

#### Acknowledgment

The authors would like to thank the institutions where they conduct their research for supporting this project and also the financial support provided by CNPq and PRP/UNICAMP.

#### References

- [1] PORTO, R. M. Hidráulica Básica. 2.ed. São Carlos. EESC-USP. 2001.
- [2] AZEVEDO NETTO, FERNANDEZ, M. F., Araújo, R., Ito, A. E. Manual de Hidráulica. 8.ed. 2002.
- [3] LAUNDER, B. E.; SPALDING, D. B. Lectures in Mathematical Models of Turbulence. London: Academic Press, England, 1972.
- [4] PATANKAR, S. V.; SPALDING, D. B. Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows. Int. J. Heat Mass Transfer. v. 15, p. 1789. 1972.
- [5] MALHOTRA, A.; BRANION, R. M. R.; HAUPTMANN, E. G. Modelling the Flow in a Hydrocyclone. Canadian J. Chem. Eng. v. 72, p. 953-960. 1994.
- [6] CULLIVAN, J. C.; WILLIAMS, R. A.; CROSS, C. R. Understanding the Hydrocyclone Separator Through Computational Fluid Dynamics. Trans IChemE. v. 81, p. 455-466. 2003.
- [7] FONTES, Carlos E.; SILVA, Luiz F. L. R.; LAGE, Paulo L. C.; RODRIGUES, Ricardo C. Introdução a Fluidodinâmica Computacional, 1. ed. Rio de Janeiro: Escola Piloto Virtual-UFRJ, 2005.
- [8] OLINO, A. L. M. Otimização do impelidor KPC utilizando Fluidodinâmica Computacional (CFD). Campinas-SP: 2010. Dissertação (Mestrado) – Universidade Estadual de Campinas.
- [9] SOARES, C. Desenvolvimento de uma metodologia para avaliação numérica e experimental do escoamento líquido/vapor em colunas de destilação. Campinas-SP: 2005. Tese (Doutorado) - Universidade Estadual de Campinas.
- [10] MALISKA C.R. Transferência de Calor e Mecânica dos Fluidos Computacional. Florianópolis: LTC, 2004.
- [11] LOMBARDI, J. C. Análise de distribuição de pressão em válvulas de diafragma poroso. 2005. 125 f. Tese (Doutorado em computação aplicada) – Instituto Nacional de Pesquisa Espaciais, São José dos Campos.
- [12] DEVENS, J.A. Quantificação do coeficiente de dispersão longitudinal em pequenos cursos d'água naturais com o uso de traçador ambientalmente neutro. Dissertação de mestrado. Universidade federal de Ouro Preto - Ouro Preto, 2006.
- [13] FLETCHER, W.J. and R.J. TREGONNING, 1992. Distribution and timing of spawning by the australian pilchard, *Sardinops sagax neopilchardus* off Albany, Western Australia. Aust. J. Mar. Freshwat. Res. 43:1437-1449.
- [14] TANNEHILL, I.R. 1947. Drought and Its Causes and Effects. Princeton University Press, 597 pp.
- [15] SIMÃO, J. A. C. Modelação em Hidráulica Fluvial e Ambiente. 2. Ed. COIMBRA. 2009.