25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

STUDY OF THE APPLICATION OF COMPUTATIONAL FLUID DYNAMICS TO EVALUATE THE OPERATING PARAMETERS OF A VENTURI FLOWMETER

Juliana Abreu de Freitas
Emerson Canton Christiano
Universidade Federal de São João del Rei
julianafreitaz@gmail.com
cantonemerson9@gmail.com

Reimar de Oliveira Lourenço
Aderjane Ferreira Lacerda
Universidade Federal de São João del Rei
reimar@ufsj.edu.br
aderjane@ufsj.edu.br

Abstract. *In the present work, simulations were performed using Computational Fluid Dynamics (CFD) techniques to evaluate the behavior of a fluid when flowing through a Venturi flowmeter. In addition, the efficiency of the pressure recovery after the tube's throat was evaluated by modifying its parameters of construction and operation. A complete factorial design 2³ was designed to statistically analyze the simulations, being the factors: velocity, convergent angle and divergent angle. Tubes with a convergence angle of 24.15º and divergent angle of 5.95º, flowing with a velocity of 3.0 m.s*$^{-1}$*, presented better results in terms of pressure recovery. The worst result was observed when working with angles of 17,85º of convergence, 8,05º of divergence and velocity of 0,1 m.s*$^{-1}$*. Thus, velocity and angle of convergence were shown to be interfering with the pressure recovery efficiency response.*

Keywords: *flowmeter, Venturi, computational fluid dynamics, factorial design*

1. INTRODUCTION

In various industrial processes, transportation by fluids is required and in that context, the fluid flows through industrial pipes. Fluid's behavior is analyzed by mechanic of fluids techniques. It involves several characteristics of the fluid of interest, hence it is needed to develop a system to describe such characteristics in a qualitative and quantitative way (Munson & Young, 2004). The pressure exerted by a fluid relates to its bombardment on a surface (White, 2011). Therefore, as it flows through a surface, it acquires a velocity and a mass flow ratio, making it necessary to develop tools for the analysis of those measures. These devices were developed from the Bernoulli equation (Ibars, 2004).

An efficient way to measure mass flow is to install a restriction to the tube and measure the pressure difference between regions of low velocity and high pressure and high velocity and low pressure (Munson & Young, 2004). Typically, differential pressure devices are used, which can be easily operated, simply constructed and at a low cost, serving for different types of fluids (Gutierrez, 2003, apud (Ibars, 2004)). Among these devices, the metering orifice, the position meter and the Venturi (Ibars, 2004) are commonly used. The choice of the best meter should take into account factors such as loss of charge and cost (White, 2011).

The Venturi meter is widely used for flow measurement in pipes, having the characteristic of being self-cleaning, preventing the accumulation of solid particles and being able to measure for fluids with high quantity of sediments, thus being used for the flow of different types of fluids in industrial processes (Dias, Silva & Filho, 2009).

The flowmeter consists of a section wide as the pipe diameter, followed by a converging conical section that leads to a cylindrical throat. Then it opens to a conic section that diverges to a cylindrical section. A piezometer ring can be coupled to the tube, between the part where the fluid inlet occurs and the restriction part, the throat, for measuring the pressure of the fluids and calculating the pressure drop (Dias, Silva & Filho, 2009).

A drawing of a venturi type flowmeter can be seen in Figure 1. In this scheme, the fluid inlet is located on the left of the image and the outlet on the right.

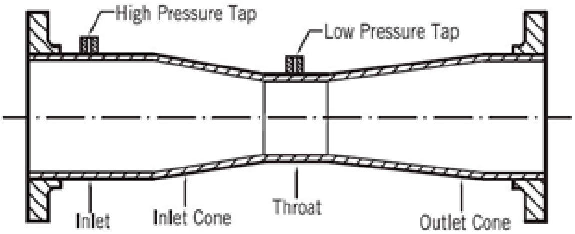

Figure 1. Drawing of venturi tube.

To analyze and predict problems on industrial processes, such as transportation by fluids, the utilization of computational fluid dynamics (CFD) is increasing. CFD is interesting to solve problems using computers to create numerical simulations based on mathematical models (Dias, Silva & Filho, 2009). It requires the construction of a geometry and then the construction of a control mesh to apply transportations equations to that control volume (Oliveira, Vieira & Damasceno, 2011). The responses of pressure collected after the simulations can be used to evaluate construction and operation parameters for the Venturi flowmeter.

1.1 Mathematical Models and Control Meshes

Computational control meshes are created by lines drawn in the control volume, which is the geometry of interest. Therefore, a mesh is a discretization (representation) of the physical plane used during the numerical simulation. When well-constructed, a system of differential equations may have a simple solution to the problem. Refining the mesh is thus necessary so that variations in physical properties of the flow, such as velocity and pressure, are best captured (Frari & Pedroso, 2009).

Two mathematical models for a high Reynolds number are proposed: κ-ε and Reynolds Stress. The κ-ε model is the most used model for computational fluid dynamics simulations, consisting of two differential equations, one for turbulent kinetic energy κ and one for the dissipation rate ε (Versteeg & Malalasekera, 2007). The Reynolds Stress model approximates the Navier-Stokes model since it calculates the transportation equations using Reynolds tensions and dissipation equation. For a three-dimensional model, as in the present study, seven equations are accounted. The greatest advantage of this model is the ability to calculate precisely complex flows, however it has a high computational cost (Alves, 2018).

Those two models were previously tested for this study, showing approximate results. Therefore, for computational costs saving, it was assumed the κ-ε model for the simulations.

2. METHODOLOGY

In this study, the operation of a Venturi type flowmeter was simulated. It was assumed that the flow occurs in steady state and there is a non-slip condition in the walls of the tube. Water at 25°C was used as study fluid, with $\rho = 998.2\ kg.m^{-3}$ and $\kappa = 0.001003\ kg.m^{-1}s^{-1}$ (White, 2011). The Venturi tube was designed in three dimensions, and sections of 30 cm were added at its end in order to represent a real pipe, as can be seen in Figure 2.

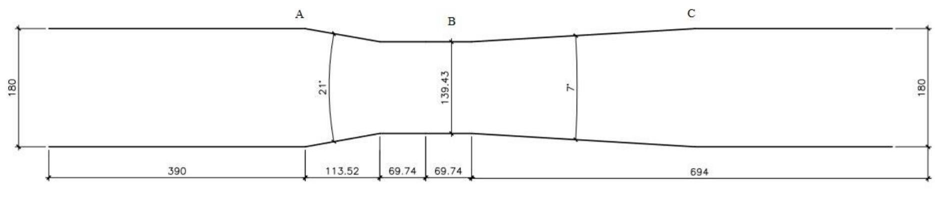

Figure 2. Venturi tube dimensions (mm) used for this study.

The objective of the computational simulation was to study different Venturi geometries for previous analysis of its construction and operation parameters. Hence, when analyzing different models of tubes, it is possible to visualize the profiles of pressure and velocity to determine which factors of construction and operation influence the pressure drop that occurs inside the tube and the recovery of this pressure at the end of the Venturi. With such information, a tube can be built with a high operating efficiency. For this study, a 2^3 (Box, Hunter & Hunter, 2005) factorial experiment was conducted with two levels of study, varying three factors: velocity (v), convergent angle (θ) and divergent angle (λ). The values chosen for the angles are based on the classic Venturi flowmeter, developed by Clemens Herschel in 1898 (Munson & Young, 2004), shifting the values by 15% up and down.

The mesh (Figure 3) was produced from tetrahedral shapes with a maximum size of 0.0085 m, counting a total of 66263 nodes and 222521 elements.

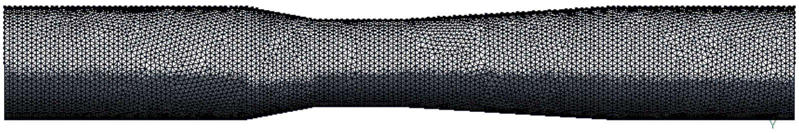

Figure 3. Illustration of the Computational Mesh.

The parameters used in the investigated Venturi simulations can be obtained from the following table.

Table 1. The simulations parameters.

Exp.	1	2	3	4	5	6	7	8	9	10	11	12	13
v($m.s^{-1}$)	0.1	3.0	0.1	3.0	0.1	3.0	0.1	3.0	1.55	1.55	1.55	1.55	1.55
θ (º)	17.85	17.85	24.15	24.15	17.85	17.85	24.15	24.15	21	21	21	21	21
λ (º)	5.95	5.95	5.95	5.95	8.05	8.05	8.05	8.05	7	7	7	7	7

3. RESULTS AND DISCUSSION

The behavior of the static pressure profile along the pipe is illustrated on Fig. 4, with v=1.55 $m.s^{-1}$, θ=7º and λ=21º (Exp. 9). At the beginning of the pipe the fluid has high static pressure, around $2.25.10^3$ Pa, and upon arriving the throat it undergoes a pressure drop, which is partially recovered in the converging section. In this experiment the recovery percentage was 86.5%.

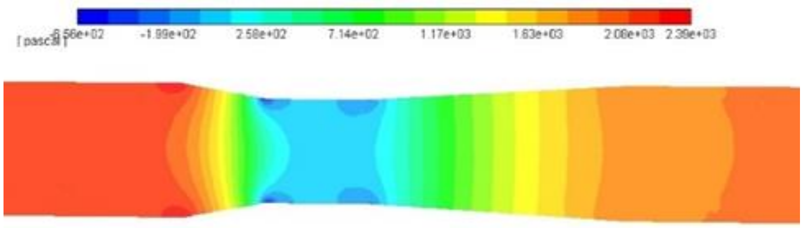

Figure 4. Static pressure profile along the pipe.

Figure 5 shows the graph representing static pressure behavior

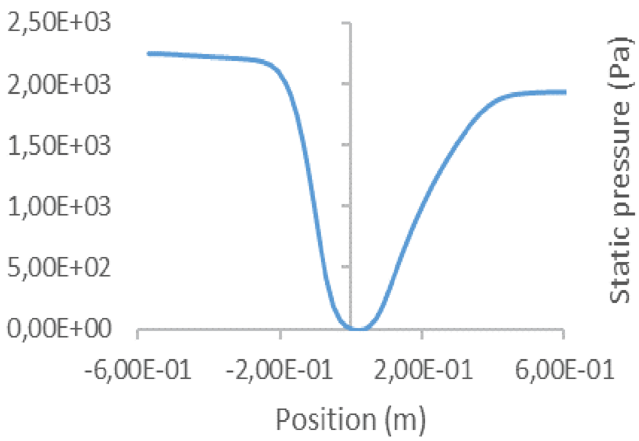

Figure 5. Variation of static pressure along flow

Figure 6 shows the behavior of the velocity profile along the pipe, with same conditions of Fig. 4. At the beginning of the pipe, the fluid has a low velocity, but when arriving in the throat region the fluid increases velocity and then loses velocity upon reaching the expansion zone. This is explained by the Bernoulli principle, since there is a conversion of pressure energy into potential energy due to strangulation in the throat zone (Munson & Young, 2004).

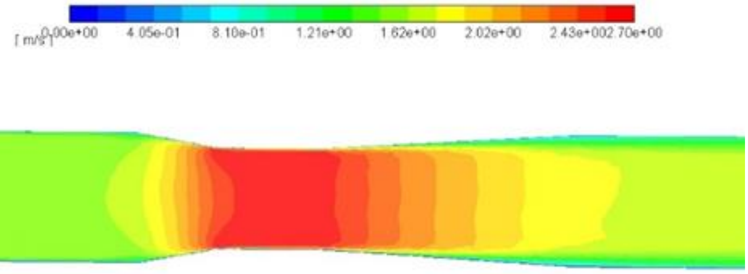

Figure 6. Velocity profile along the pipe.

Analysis of the vector behavior of the fluid during its passage through the fluid is made and it is arranged in Figure 7, then Figures 8 and 9 show a cutout in the throat region and in the vicinity of the tube, respectively.

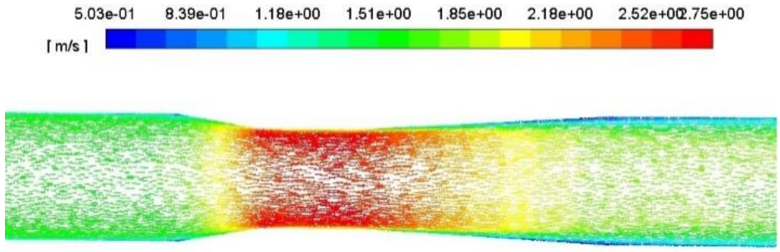

Figure 7. Velocity field vectors.

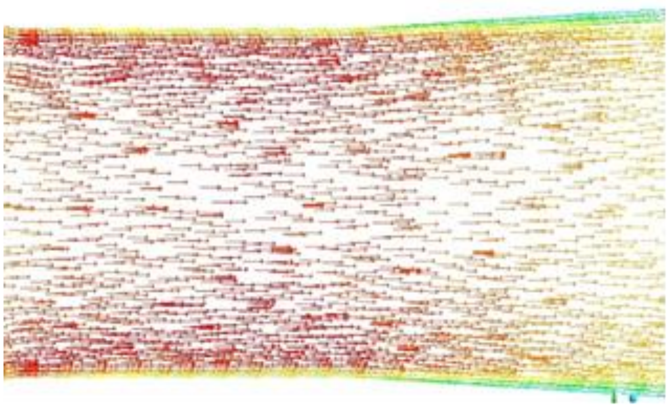

Figure 8. Velocity field vector in the throat region.

Figure 9. Fluid vector behavior near pipe.

From the analysis of vector behavior it is possible to notice that there is no formation of vortices and that the behavior of vectors is not chaotic, this is due to the fact that the components of the Venturi tube are relatively simple and there is no presence of large impediments or sudden variations in direction.

In the throat it is possible to observe a tendency of the accumulation of vectors, since in the region occurs the process of strangulation of the fluid and the consequent increase of its velocity. In the vicinity of the wall, the velocity of the vectors decreases in this region, there is a smaller accumulation of vectors, and it is possible to perceive more explicitly the effects of the boundary layer in the region.

For each simulation, static pressure measurements were taken at the beginning of the tube (A) and at the end of the tube (C) to extract the pressure variation data within it, and graphs were obtained for each case (Figures 10 and 11)

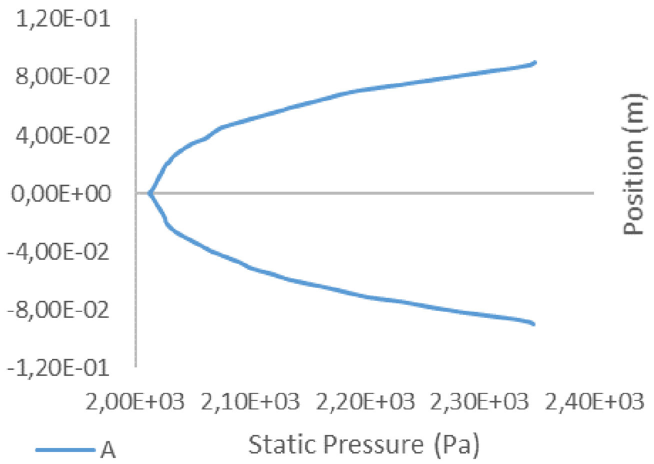

Figure 10. Variation of static pressure along region A of pipe length: $v = 1.55 m.s^{-1}$, $\theta = 21^{o}$ and $\lambda = 7^{o}$

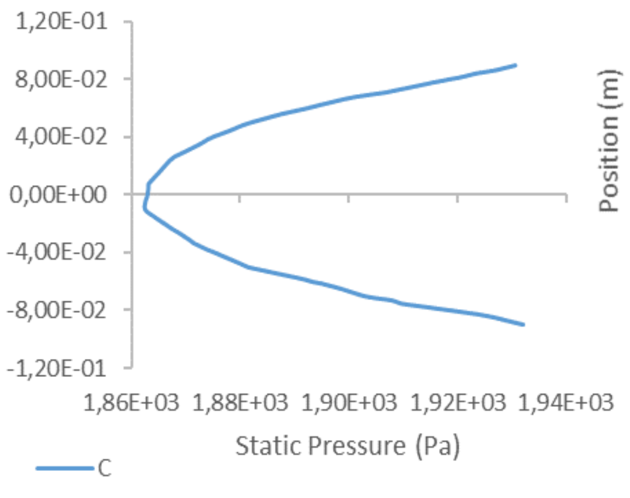

Figure 11. Variation of static pressure along region C of pipe length: $v = 1.55 m.s^{-1}$, $\theta = 21^{o}$ and $\lambda = 7^{o}$

Pressure recovery can be calculated from Equation 1:

$$\text{Recovery}\ (\%) = \frac{P_c}{P_a} * 100\%$$

Where Pc symbolizes the pressure take-off at the end of the divergent region and Pa symbolizes the pressure take-off at the beginning of the convergent region.

Using Equation 1 as the basis, pressure recovery was calculated for each experiment. The calculated results are described in Table 2

Pressure recovery (ΔP) was measured for each of the experiments and the values are described in Tab. 2.

Table 2. Results obtained for the pressure recovery for each condition.

Exp.	1	2	3	4	5	6	7	8	9	10	11	12	13
v($m.s^{-1}$)	0.1	3.0	0.1	3.0	0.1	3.0	0.1	3.0	1.55	1.55	1.55	1.55	1.55
θ (º)	17.85	17.85	24.15	24.15	17.85	17.85	24.15	24.15	21	21	21	21	21
λ (º)	5.95	5.95	5.95	5.95	8.05	8.05	8.05	8.05	7	7	7	7	7
Pressure Recovery	77.90	89.92	84.35	97.01	75.74	86.05	80.84	93.09	82.30	82.30	82.30	82.30	82.30

The biggest recovery in pressure was at Exp. 4, with v=3.0 $m.s^{-1}$, θ=24.15º e λ=5.95º. The second highest value was at Exp. 8, with v=3.0 $m.s^{-1}$, θ=24.15º e λ=8.05º. It can be seen that both conditions have the same value of the factors v and θ, differing from 3.92%, what can be justified because of the divergent angle variation. The worst value of pressure recovery is in Exp. 5, with v=0.1 $m.s^{-1}$, θ=17.85º e λ=8.05º. and the second worst value is in Exp. 1. with v=0.1 $m.s^{-1}$. θ=17.85º e λ=5.95º. The experiment shows that for low velocities and low convergent angles the pressure recovery decreases.

In addition, divergent angle has little influence on the results, although its increase leads to a lower recovery rate. Santos & Béttega (2015) also analyzed experiments with contraction angles of 16.8º through 25.2º and expansion angles from 12º to 18º, in six different velocities, resulting on low pressure recovery for low velocities and low convergent angle, as well as in this study.

Pareto chart (Fig. 12) was generated from the values obtained for the study of influence of each variable in the pressure recovery of the fluid flowing through a Venturi flowmeter.

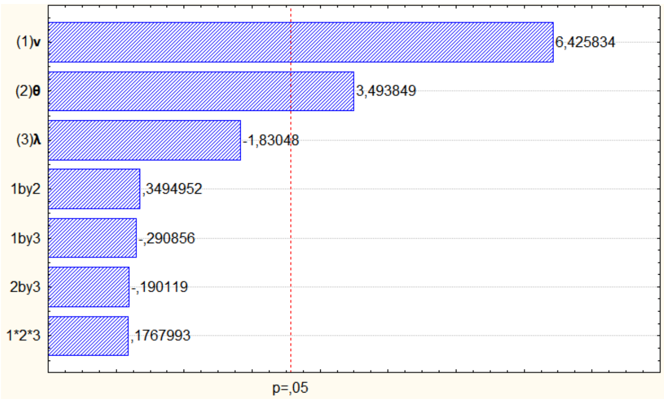

Figure 12: Pareto chart.

Figure 12 shows that velocity has the highest influence on pressure recovery. Followed by the convergent angle. The divergent angle had no influence on the answers. as well as the combination of the variables. The analysis of this figure confirms the results obtained in the best and worst experiments since higher values of velocity and convergent angle result in higher rates of pressure recovery whilst smaller values of velocity and convergent angle result in worse rates of recovery of pressure. This shows that these factors are important during the construction and operation of a Venturi flowmeter. therefore that can optimize before construction for a better operation. The divergent angle variation had very little impact on pressure recovery rate. which is confirmed by the Pareto chart. in which the analysis of that variable has no statistic interference on the results.

4. CONCLUSIONS

Computational fluid dynamics simulation is very useful for solving problems as it reduces human effort and can predict different conformations of a design prior to its construction. From the simulations. it was possible to analyze the velocity and pressure profiles of a water flow in the Venturi flowmeter accurately. Therefore. it was possible vary in the parameters of construction and operation of the Venturi to study their impacts on the pressure recovery rate.

Observing the results obtained for different conformations at different velocities it was possible to choose the best conformation that presented a convergent angle of 24.15º and divergent angle of 8.05º, operating at a velocity of 3.0 $m.s^{-1}$. in which a value of 97.01% pressure recovery was obtained at the end of the tube. The worst experiment had a recovery value of 75.74%, with angles of 17.85º and 8.05º for the contraction and expansion sessions, respectively, while operating at a velocity of 0.1 $m.s^{-1}$.

Through the statistical analysis of the response data of pressure recovery it was possible to conclude that the factors that most influence on the response are the fluid flow velocity and the angle of convergence. The divergent angle did not present significant interference between the analyzed range of 5.95° to 8.05°. There was also a tendency to improve pressure recovery by increasing the velocity and angle of convergence. Future studies on those parameters are required for better conclusions.

5. REFERENCES

Alves. B. A. Simulação de um trocador de calor casco e tubos com fluido turbulento utilizando CFD. Universidade Tecnológica Federal do Paraná. **2018**.

Box. G. E. P.; Hunter. J. S.; Hunter. W. G. in *Statistic for Experimenters*. Wiley-Interscience. 2 ed. United States of America. **2005**.

Dias. A.; Silva. F. G. B.; Filho. G. L. T. Estudo da Distribuição da Velocidade em Tubo Venturi Utilizando Medidas Experimentais e Técnicas de CFD. *Revista Brasileira de Recursos Hídricos*. **2009.** *14*. 81-92.

Frari. R. V.; Pedroso. C. A. Malhas Computacionais para Simulação Numérica de Escoamentos de Fluidos Entre Cilindros com Excentricidade. X Encontro Gaúcho de Educação Matemática. Ijuí. **2009**.

Ibars. R. A. F. Desenvolvimento e Avaliação de Tubos Venturi para Medição de Vazão. Master's Thesis. Universidade de São Paulo. 2004.

Munson. B. R.; Young. D. F.; Okiishi. T. H. in *Fundamentos da Mecânica dos Fluidos*. translated by E. J. Zerbini. Edgard Blücher. São Paulo. **2004**.

Oliveira. N. M. B.; Vieira. L. G. M.; Damasceno. J. J. R. Fluidodinâmica Computacional Aplicada ao Estudo de Medidores de Vazão. *Horizonte Científico.* **2011**. *5*.

Santos. M. G. P.; Béttega. R. Avaliação dos Parâmetros de Projeto de Medidores de Vazão tipo Venturi Utilizando Fluidodinâmica Computacional. XI Congresso Brasileiro de Engenharia Química em Iniciação Científica. Campinas. **2015**.

Versteeg. H. K.; Malalasekera. W. An Introduction to Computational Fluid Dynamics: The finite volume method. 2 ed. England. Pearson Education Limited. **2007**.

White. F. M. in *Mecânica dos Fluidos*. translated by Fecchio. M. M.; Filho. N. M. AMGH. Porto Alegre. **2011**.

6. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.