NUMERICAL STUDY OF THE TURBULENT HYDRAULIC FLOW IN VENTURI TUBES FOR DIDACTIC PURPOSES IN FLUID MECHANICS

Rabello, L. L.

Gava, W. V. D

Furieri, B.

Lovatte, E. R.

Caliman, M. C. S. F.

eng.liviarabello@gmail.com

walquiria.vdgava@gmail.com

bruno.furieri@ifes.edu.br

enilene.lovatte@ifes.edu.br

claracaliman@gmail.com

Instituto Federal de Educação, Ciência e Tecnologia do Espírito Santo

Avenida Vitória, 1729, 29040-780, Vitória – ES, Brazil

Abstract. *One of the most used instruments for flow measurement in hydraulic investigation is the Venturi tube, which is based on a very simple technique, reducing the area of passage of the flow, which consequently results in an increase in velocity magnitude and turbulence. Venturi tubes are advantageous because they are self-cleaning and therefore have the characteristic of being used to measure flow rate with large amount of sediment, which is a recurrent condition in environmental flows, thus justifying the relevance of this study to several areas of engineering, specially water resources and sanitation. The work concentrated on investigating the flow inside these structures as part of Fluid Mechanics in undergraduate courses. In this, students are presented, besides the mathematical theory and equation necessary to understand the problems, the influence of discretization, numerical method and turbulence models in results. The present work has the general objective to evaluate, through computational modeling, Venturi tubes designed with the CFD (Computational Fluid Dynamics) tool using ANSYS Fluent software. In order to identify the greatest number of parameters and variables essential to the understanding of the phenomena involved in the flow*

Proceedings of the XXXVIII Iberian Latin-American Congress on Computational Methods in Engineering
P.O. Faria, R.H. Lopez, L.F.F. Miguel, W.J.S. Gomes, M. Noronha (Editores), ABMEC, Florianópolis, SC, Brazil, November 5-8, 2017.

in Venturi tubes the following points were addressed in tested cases: (i) variation in the geometry of the pipe strangulation section, (ii) refinement of the mesh (Iii) turbulence modelling and (iv) methods of spatial interpolation. A qualitative analysis of the tool indicated that it is interesting to facilitate the understanding of the complex equations and theories involved in the prediction of turbulent flows and the use of numerical methods. The main numerical results obtained showed the strong influence of mesh refinement in the central region of the Venturi tube, as well as its correct association with the appropriate turbulence model. The computational tool has proven useful in checking recirculation zones that can become sediment accumulation when applicable.

Keywords: *Numerical methods, Venturi tube, Fluid Mechanics, Hydraulics, Turbulent flow*

1 INTRODUCTION

The Venturi is a device that was used initially to solve simple problems and improve knowledge about hydraulics. However, years after its development, it began to be used in industrial applications, mainly, for flow measurement. Today, the Venturi principle is used for various technological purposes in numerous industrial applications, such as air purifiers, gas-solids injectors and jet lift pumps. Industrial applications of Venturi tubes range from small, non-engineered adjustments to more complex engineering adaptations (Lima & Caetano, 2015).

Computational Fluid Dynamics (CFD) is a modeling tool used to generate flow simulations. It numerically solves the governing laws of Fluid Dynamics (namely conservation of mass and momentum).

CFD was used to gain insight in the aerodynamic performance of a Venturi-shaped roof (called VENTEC roof), which the simulations were performed with the 3D steady Reynolds-averaged Navier–Stokes (RANS) equations and the k–e model. The CFD simulations were based on a detailed grid-sensitivity analysis and on successful validation of the grid-independent results by comparison with experiments in an atmospheric boundary layer wind tunnel. The study illustrated the use of CFD to increase insight in building aerodynamics and to support sustainable building design (Blocken et al., 2011).

Ali et al., 2012 used CFD simulations to predict pressure drop in Venturi scrubber. ANSYS CFX tool was used to obtain the simulation results. The k-ε turbulence model was employed to study the fluid dynamics of the Venturi scrubber. In this case, Mesh size played an important role for convergence and accuracy of results. Therefore, the mesh independency was checked for optimized mesh size.

Dehkordi et al., 2017 simulated a multiphase flow using ANSYS Fluent. They characterize core-annular flow patterns of highly-viscous oil-water flows through a Venturi Flow Metter (VFM) and a Nozzle Flow Metter (NFM) in a 40 mm i.d. horizontal pipe. Eight cases were considered for numerical runs and two turbulent models (k–e and k–w). Two phase pressure drop, instantaneous radial velocity and holdup profile, cross-sectional time-averaged holdup and slip ratio were extracted from CFD simulations. Flow patterns were also compared to actual images taken by a Olympus E-M10 mirrorless camera. In terms of concentrated pressure drop and mean water holdup, the results of CFD simulations were consistent with the experimental data. Prediction of the total mass flow rate, computed by homogeneous model, for both the VFM and NFM also gave very satisfactory results.

Ashrafizadeh & Ghassemi, 2015 investigated Cavitating Venturis devices which can be used to passively control the flow rate of fluids. The study used experimental and numerical investigation. The numerical results showed agreement with the experimental data by a maximum deviation of 5–10% and confirmed that the numerical approach can be used to predict the critical pressure ratio and mass flow rate at cavitating condition.

CFD can be used for scientific purposes, but also has great interest from an educational point of view. These software packages have great importance when results are contrasted to experiments performed in the laboratory. Students can visualize magnitudes obtained by the computational model, which might be compared with the theory of the object under study. Therefore, it is possible to achieve the understanding of fluid movement phenomena (Lopez et al., 2008).

Polytechnic University of Cartagena in Spain used CFD in a course on Selected Topics of Fluid Mechanics course for the Industrial Engineer degree, employing numerical techniques as a motivating element for students. The main objectives of the course is to provide an understanding of the analytical methods employed for selected topics in Fluid Mechanics and to bring a familiarity with some experimental and numerical techniques. To improve teaching-learning, a pedagogical method based on Problem-Solving Based Learning was applied. As a result, the motivation and the theoretical and technical abilities of the students had been improved. It could be concluded that the method was efficient and had increased the students' satisfaction (Zamora & Kaiser, 2009).

The present paper presents simulation of two different geometries of Venturi tube for hydraulic investigation purpose, using a turbulent model called Standard k-epsilon, which allows the analysis of this device in several engineering applications. This work is especially relevant for the water resources area, which is part of the curriculum of Sanitary and Environmental Engineering, since it provides measures of the flow rate with large amounts of sediment, quite common in Brazilian rivers. Despite of the technical applications of the present numerical simulations it is proposed a methodology of using CFD as a didactic tool in undergraduate and graduate courses.

2 NUMERICAL SIMULATION METHODOLOGY

For the present study, two different geometries for Venturi tube were designed in the ANSYS Workbench 17.2 – *Student version*. The numerical experiment has been designed to allow students to identify and learn easily the main physical concepts involved in this kind of tool. Figure 1 and Figure 2 present the two distinct geometries, called Case A and Case B. The main differences are noticed in the dimensions of the Venturi itself, it means, the convergence part.

The Venturi tube of Case A has a larger cylindrical tube between the convergent and divergent (as the most part of Venturi tubes) than Case B. For both cases, the inlet velocity profile was set uniform and its magnitude is 1.5 m/s. The fluid is water and the following hypothesis were set: isothermal, Newtonian and incompressible.

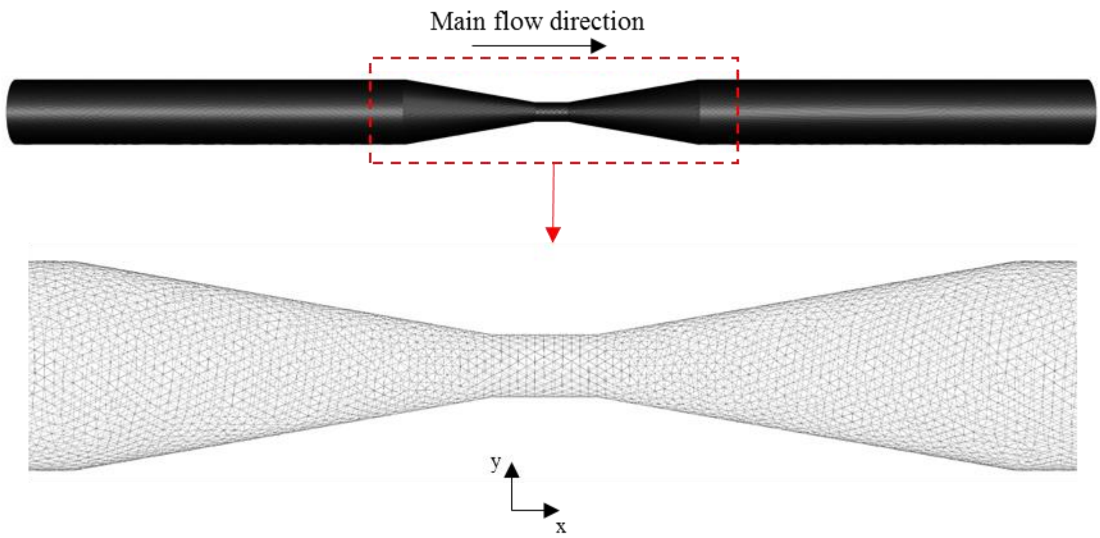

Figure 1 – Computational domain and numerical mesh of Case A

The Venturi tube of Case B does not present a link between divergent and convergent. It works almost like a hole inside the tube. The design of each case was thought to be very different scenarios. It is important for engineering students to understand the implications of a well-executed project. Additionally, very opposite situations carry out the students to think more detailed about every situation.

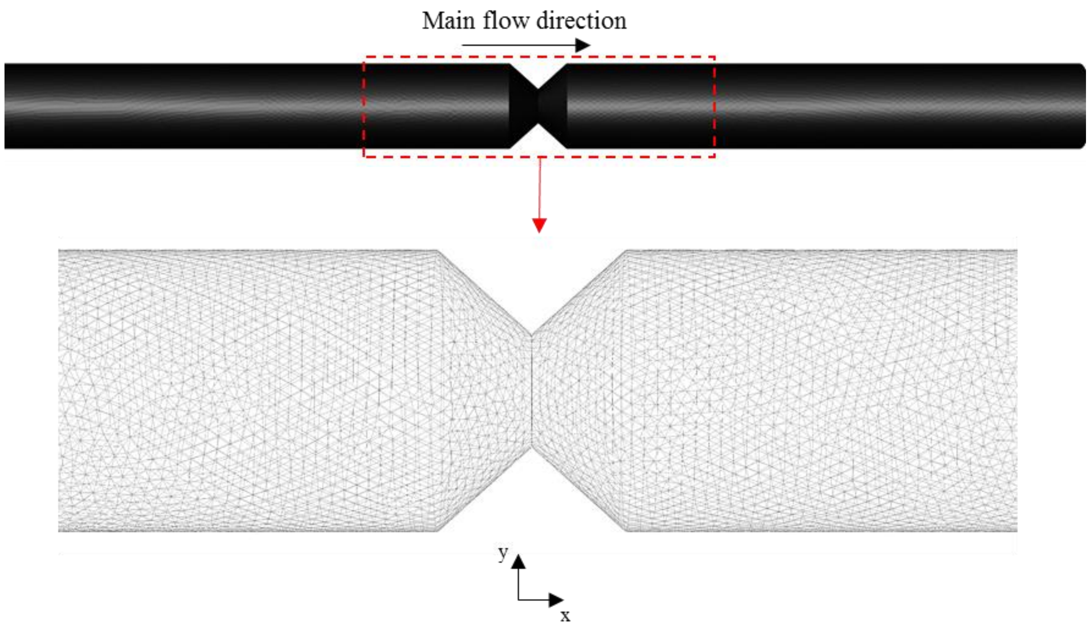

Figure 2 – Computational domain and numerical mesh of Case B

Another very important step in doing numerical simulations is the computational mesh. The students built the mesh in order to understand how it affects the final result of a numerical simulation. The main point noticed were the refinement very close to the walls and where it was observed the most adverse gradients of all variables. The following sub-sections present, briefly, how the authors and students dealt with each step in a numerical simulation.

2.1 Governing equations of fluid dynamics

The turbulent flow are challenging since they have unsteady aperiodic motion and contain a wide range of scales, so they strongly depend on initial conditions.

The computational model solves the differential equations that represent the flow and produces solutions for the velocity and pressure fields in each node of the mesh.

The governing equations for a turbulent flow are Eq. 1 and Eq. 2:

Mass Conservation

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

Momentum

$$\frac{Du_i}{D_t} = g_1 - \frac{1}{v}\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_i} \tag{2}$$

They are partial differential equations that allow determining the velocity and pressure fields.

When mass conservation is expressed mathematically in a fluid flow, the continuity equation is obtained, which, in Cartesian geometry, is represented by the following partial differential equation (Eq. 3):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}.(\rho u) + \frac{\partial}{\partial y}.(\rho v) + \frac{\partial}{\partial z}.(\rho w) = 0 \tag{3}$$

where ρ is the specific gravity, t is the time, $(x, y\ , z)$ are the three Cartesian coordinates, and (u, v, w) are the three components of the velocity vector corresponding to the directions x, y and z, respectively (Çengel & Cimbala, 2010).

In the used CFD technique, the governing transport equations is discretized using the finite volume method (FVM).

This method is based on the realization of mass, momentum and/or energy balances on a given control volume, where the flows of the variables cross the faces of the volume. The main advantage of the FVM is that the spatial discretization is carried out directly in the physical space. Thus, there are no problems with any transformation between coordinate systems. Since the FVM is based on the direct discretization of the conservation laws, mass, momentum and energy are also conserved by the numerical scheme. This leads to another important feature of the method, namely the ability to compute weak solutions of the governing equations correctly. (Blazec, 2015).

2.2 Numerical mesh

Once the geometries and the names of the contour surfaces have been defined, the meshes were created for each geometry. The boundary conditions are very important because solid walls are the main source of vorticity and turbulence. In addition, the mesh was refined in the Venturi tube stricture region and in the fluid inlet and outlet regions to better visualize the fluid flow.

After mesh generation, we proceed to the pre-processing, where the type of regime is defined as turbulent followed by the turbulence model.

2.3 Turbulence modelling

The computational model is based on solutions of Navier-Stokes equations for incompressible flows, using a finite volume method of non-uniform structured mesh. The Navier-Stokes equations (mass conservation and momentum) are able to describe the flow for both a laminar and a turbulent regime.

The main drawback of the k one-equation model is the incomplete representation of the two scales required to build the eddy viscosity. The two-equation models try to represent both scales independently.

In this work, we used a two-equation model, analyzing a turbulent flow regime in the Standard k-epsilon model.

The basis for standard k-ε model is the Boussinesq hypothesis, which postulates that the Reynolds stress is related linearly to mean rate of strain, directly analogous to the relation for the viscous stress in a Newtonian fluid, and the proportionality factor is eddy viscosity. The Reynolds stress tensor can be written as in Eq. 4:

$$\tau_{ij}^{R} = -\rho\ \overline{u'_i u'_j} = \mu_1 \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right) - \frac{2}{3}\rho k \delta_{ij} \ldots \tag{4}$$

where k is the turbulent kinetic energy ($k \equiv \left(\frac{1}{2}\right)\overline{u'_i u'_j}$), and μ_1 stands for the eddy viscosity, defined as $\mu_1 \equiv \rho C_\mu k^2/\varepsilon$ with ε ($\varepsilon \equiv \frac{\mu_1}{\rho}\left(\overline{\frac{\partial u_i \partial u_i}{\partial u_j \partial u_j}}\right)$) being the turbulent kinetic energy dissipation rate. Consequently, the eddy viscosity contains two unknown variables k and ε. It is necessary to provide the transport equations for these two variables derivable from the momentum equation, represented by Eq. 5 and Eq. 6:

$$\rho \frac{\partial(\bar{u}_i k)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_1}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G - \rho\varepsilon \ldots \quad (5)$$

$$\rho \frac{\partial(\bar{u}_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_1}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_1 \frac{\varepsilon}{k} G - C_2 \frac{\varepsilon^2}{k}\rho \ldots \quad (6)$$

where $G = \mu_1 \frac{\partial u_i}{\partial x_i}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$, and the constants are represented in the Table 1 (Zhang et al., 2011).

Table 1. Constants from the momentum equation

Constant	Value
C_1	1.44
C_2	1.92
C_μ	0.09
σ_k	1.0
σ_ε	1.3

2.4 Solver set-up

The solver is the component that solves the computational problem, producing the necessary results. The partial differential equations are integrated over all control volumes in the region of interest, which corresponds to applying the conservation law for each control volume. The integrated equations are converted into a System of algebraic equations to generate a set of approximations for the terms in the integral equations and then the algebraic equations are solved through numerous iterations. The solver produces a result file, which is passed to post-processing.

2.5 Convergence criteria

During simulation course, it was observed the variation of the values of these properties and when they stabilized. It was assumed that the steady state had been reached. The data used in the analyzes were all collected in the last iteration of each simulation. It was used a convergence criteria of 10^{-4} for all governing and turbulence modelling equations.

2.6 CFD as an educational tool

The main purpose of the paper was to show how CFD can be a very useful tool in engineering schools, not only inside research centers. The software must be employed as an obligatory part of Fluid Mechanics lectures. Difficult differential equations (as Navier-Stokes) are very hard to be understood without any applications and is it not viable to solve it directly.

3 RESULTS

The following images are the main graphical results obtained in the present paper. These results arose from a series of numerical simulations performed during an undergraduate subject of Fluid Mechanics.

Comparing the Static Pressure of Case A (Figure 3) and Case B (Figure 7) we noticed that in B the head loss is much more important. Students were aware of this result as they had already learned the theory of these tools that measure pressure. However, the physical concept is easily applied in other situations if an experiment is shown to them. In this case is a numerical experiment. An engineering student that has this tool (CFD) in its knowledge is able to perform any test or improve any project that involves fluids. The head loss is very important in this kind of equipment, it means, energy costs is highly dependent on this variable.

Figure 4 and Figure 8 present the results for the velocity magnitude distribution. For a Fluid Mechanics student, these graphics are the most straight interpretation of the Bernoulli Equation if analysed together with static pressure information. The acceleration noticed in Case A is much more important than Case B. Depending on the Venturi tube material it may be a problem as some materials has a limit of maximum velocity.

The turbulent kinetic energy (TKE) is the indication of how the turbulent structures are distributed in a flow. It is worth remember that in the present paper the turbulence modelling applied was the most simple existent in the software. The main reason being the computational costs and mesh refinement required by more robust models. As we used a student version mesh is limited and so the turbulence model. Because of this limitation the results of TKE must be interpreted carefully as it may be over or underestimated from the actual result. However, it is a very good qualitative result for the comparison between Case A and Case B. In Case B, the high values of TKE reach more far distances downstream the Venturi tube in comparison with Case A. Very high levels of turbulence in a hydraulic pipe may lead to the increase of shear stress and possibly damages to the material (see Figure 10 to notice that).

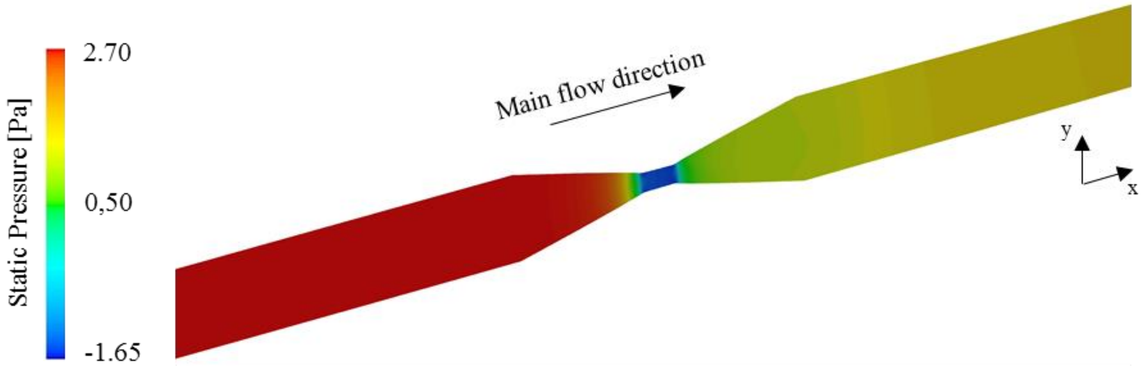

Figure 3 – Distribution of static pressure in a longitudinal plane of Case A

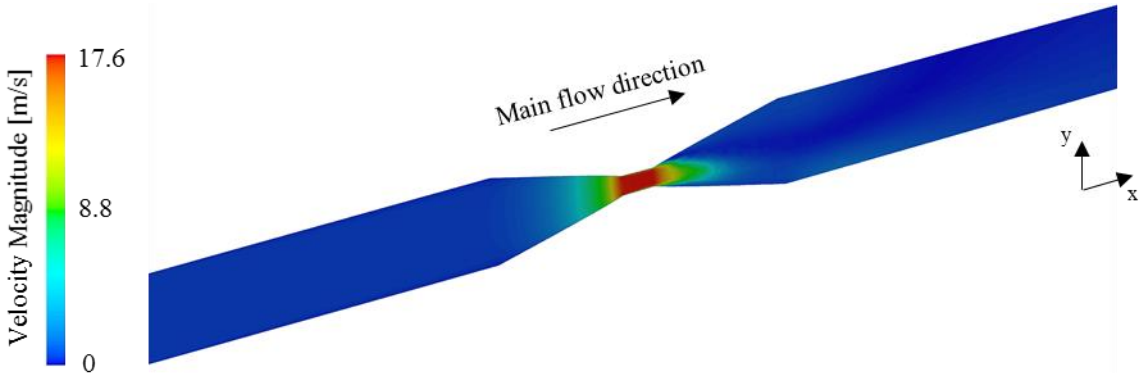

Figure 4 – Distribution of velocity magnitude in a longitudinal plane of Case A

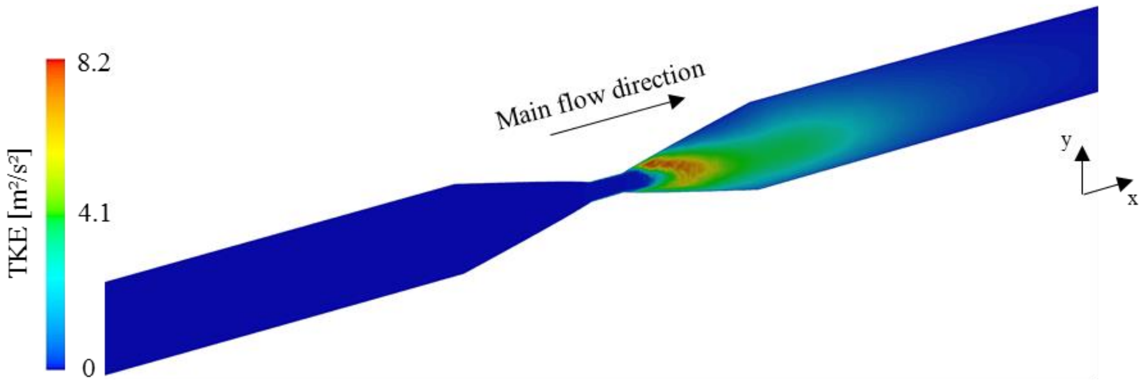

Figure 5 – Distribution of turbulent kinetic energy in a longitudinal plane of Case A

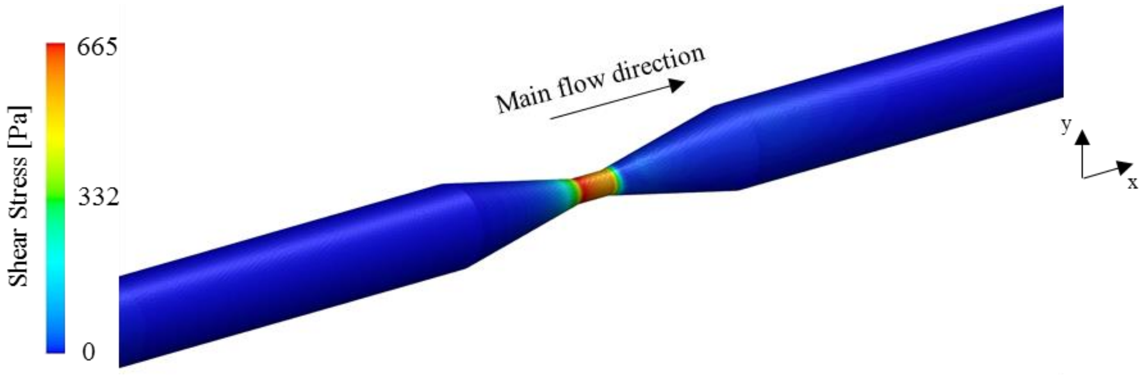

Figure 6 – Distribution of wall shear stress on the lateral walls of the Venturi tube of Case A

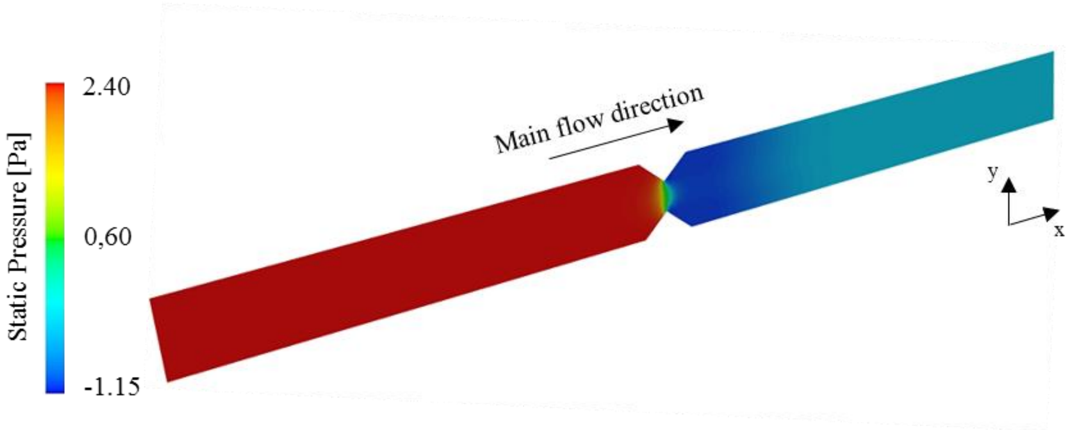

Figure 7 - Distribution of static pressure in a longitudinal plane of Case B

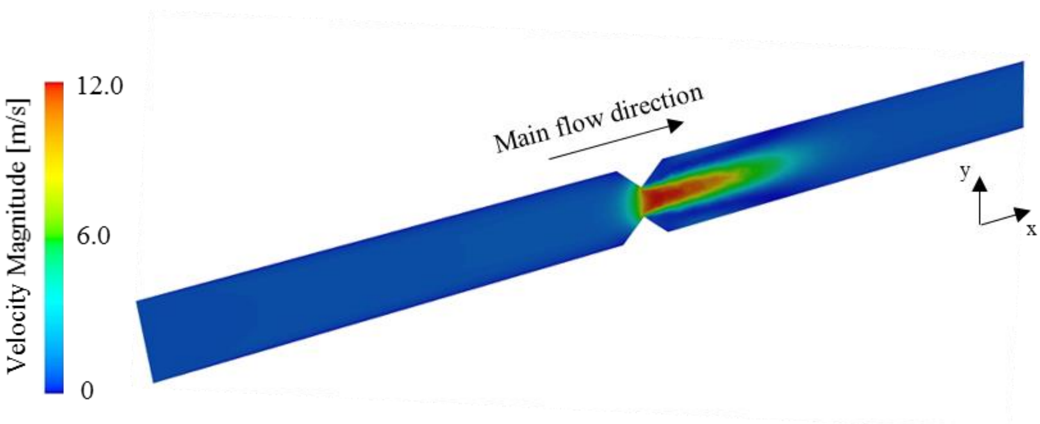

Figure 8 - Distribution of velocity magnitude in a longitudinal plane of Case B

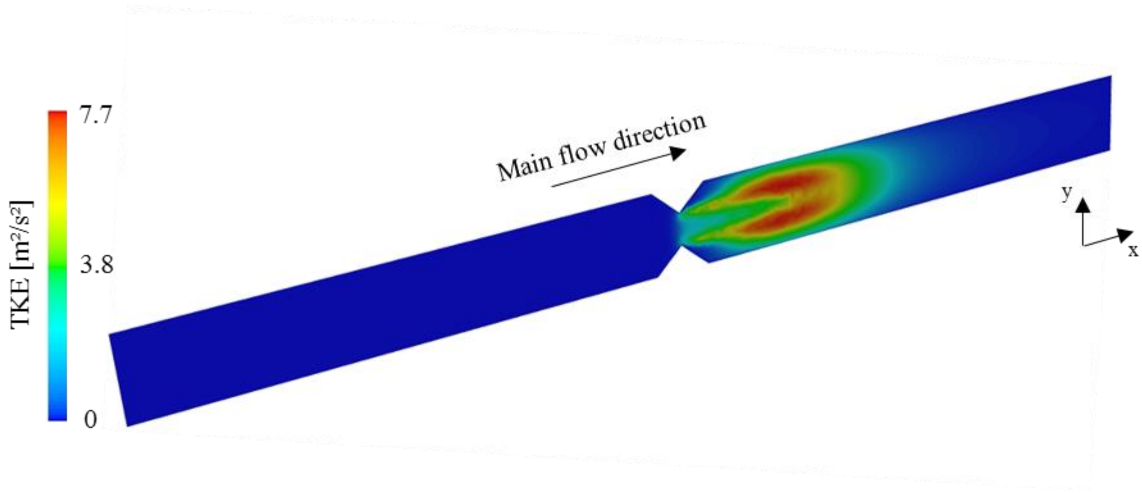

Figure 9 - Distribution of turbulent kinetic energy in a longitudinal plane of Case B

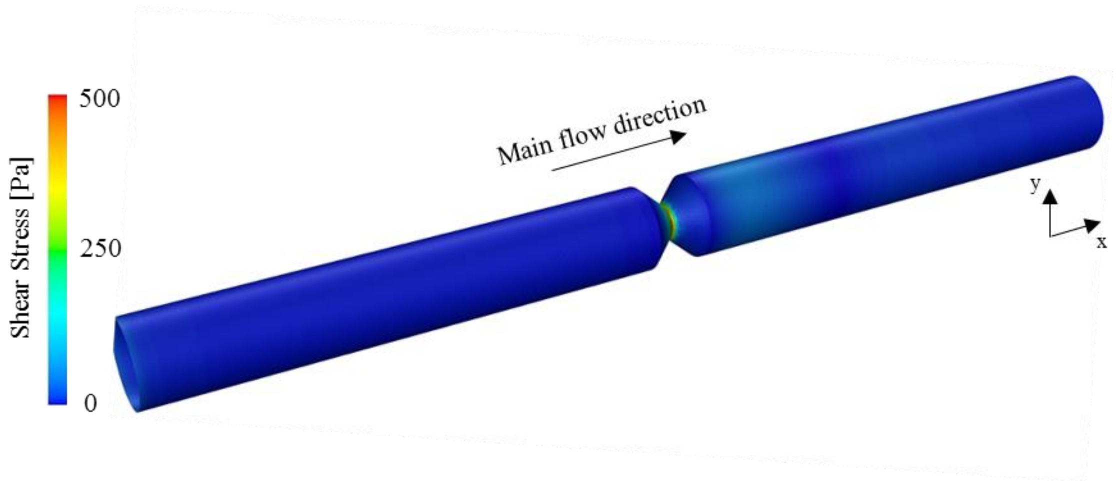

Figure 10 - Distribution of wall shear stress on the lateral walls of the Venturi tube of Case B

4 CONCLUSIONS

The present paper proposed the numerical simulation of a hydraulic equipment by Sanitary and Environmental Engineering students in order to observe the importance of this tool in their learning process. The paper writing and results discussions reflect the interpretation of an undergraduate student after learning and executing this CFD software.

Several papers are devoted to simulating and discuss technical results concerning fluid dynamics. However, less works is interested in the investigation of the teaching-learning dichotomy so important in engineering schools. By using the proposed software (or, of course, a similar software) the student is able to deal with many computational fluid mechanics concepts: (i) variables and units as pressure, velocity, shear stress and turbulent kinetic energy, (ii) computational domain and mesh refinement, (iii) turbulence and its modelling and (iv) graphical visualization.

The numerical simulations performed for the present study worked well since the theoretical behaviour presented by the literature was easily observed in every variable distribution discussed. Therefore, even a very simple numerical simulation (that required less than a lecture time to be achieved) is able to represent the real cases of hydraulic pipes.

ACKNOWLEDGEMENTS

The authors thank the Federal Institute of Espirito Santo (Ifes) for the support in doing this work.

REFERENCES

Ali, M., Changqi, Y., Zhonging, S., & Jianjun, W., 2012. CFD Simulation of Prediction of Pressure Drop in Venturi Scrubber. *Applied Mechanics and Materials,* vol. 166–169, pp. 3008–3011.

Ashrafizadeh, S. M. & Ghassemi, Hojat., 2015. Experimental and numerical investigation on the performance of small-sized cavitating Venturis. *Flow Measurement and Instrumentation,* vol. 42, pp. 6–15.

Blazek, J., 2015. *Computational Fluid Dynamics - Principles and Applications*. 3rd. Elsevier Science.

Blocken, B., Hoof, T. van, Aanen, L., & Bronsema, B., 2011. Computational analysis of the performance of a venturi-shaped roof for natural ventilation: Venturi-effect versus wind-blocking effect. *Computer & Fluids,* vol. 48, n.1, pp. 202–213.

Cengel, Y. A., Cimbala, J. M., 2010. *Fluid Mechanics: Fundamentals and applications*. McGraw-Hill.

Dehkordi, P. B., Colombo, L. P. M., Guilizzoni, M., & Sotgia, G., 2017. CFD simulation with experimental validation of oil-water core-annular flows through Venturi and Nozzle flow meters. *Journal of Petroleum Science and Engineering,* vol. 149, pp. 540–552.

Lima, L. E. M., & Caetano, N. M., 2015. Simulação computacional de um escoamento turbulento de um fluido incompressível em um Venturi. *Linkania,* vol. 5, n. 1, pp. 91–109.

Lopez, P. A., Mora, J. J., Martínez, F. J., & Izquierdo, J., 2008. Computational Fluid Dynamics (CFD) Models in the Learning Process of Hydraulic Engineering. *Computer applications in engineering education,* vol. 18, n.1, pp. 252–260.

Scott-Pomerantz, C. D., 2004. *The k-epsilon model in the theory of turbulence.* PhD thesis, University of Pittsburgh. Available in: http://d-scholarship.pitt.edu/10241/1/cdsp-new.pdf

Zamora, B. & Kaiser, A. S., 2009. Teaching Experience on Selected Topics of Fluid Mechanics using Computational Fluid Dynamics. *Formación Universitaria,* vol. 2, n.1, pp. 27–39.

Zhang, X., Jin, X., Wang, Y., Deng, K., & Ren, Z., 2011. Comparison of Standard k-ε Model and RSM on Three Dimensional Turbulent Flow in the SEN of Slab Continuous Caster Controlled by Slide Gate. *ISIJ International*, vol. 51, n. 4, pp. 581–587.