Analysis of velocity fields in a hydraulic curve long distance using computational fluid dynamics

SANTOS, I. F.[1] SILVA, F.G.B [2],BARROS,R.M.B [3], FILHO,G.L.T.F[4]

[1]UNIFEI- Instituto de Recursos Naturais, Federal University Itajubá, P.O. Box 37500-903,City, Itajubá; (55) 35-36291485; FAX (55) 35-36291265; email: ivanfelipedeice@hotmail.com

[2] NUMMARH- Research Center, Institute of Natural Research, Federal University Itajubá, P.O. Box 37500-903,City, Itajubá; PH (55) 35-36291265; FAX (55) 35-36291265; email: ffbraga.silva@gmail.com

[3]NUMMARH- Research Center, Institute of Natural Research, Federal University Itajubá, P.O. Box 37500-903,City, Itajubá; PH (55) 35-36291265; FAX (55) 35-36291265; email: remambeli@hotmail.com

[4] NUMMARH- Research Center, Institute of Natural Research, Federal University Itajubá, P.O. Box 37500-903,City, Itajubá; PH (55) 35-36291265; FAX (55) 35-36291265; email: tiago@unifei.edu.br

ABSTRACT

This article analyzes the velocity fields were performed on a hydraulic curve long distance using tools of computational fluid dynamics. Curves are accessories widely used in hydraulic circuits and act by changing the flow direction, which imposes on it, shakes accompanied by a loss of singular charge. Special features of the flow, such as disturbances of the velocity vectors, the velocity gradient between the ends of the curve and regions of different velocities of flow in the bend section could be observed by means of detailed simulation results mode.

Keywords – Fields of velocity, hydraulic radius curve long, Computational Fluid Dynamics and singular loss of load.

1. **Introduction**

The curves connections are widely used in various installations for the transport of water. These produce localized losses due to change in flow direction. By inertia efect, the fillets tend to retain their rectilinear motion and are prevented by the solid boundary of the connection. This change in direction causes a substantial change in the velocity profile and thus the pressure distribution, so that there is increased pressure on the outside of the curve with decreasing speed, and the opposite inside of the curve, which generates a spiraling motion of the particles, which persists for a considerable distance downstream of the bend. (Porto, 2006).

The spiral movement of the particles is due to the detachment of the flow from the wall. This propensity leads to detachment fluid to drive up due to inertia, against the outer wall of the bend, emerging a transverse secondary flow to the main flow. This leads to the formation of regions of separated flow that reduce the section of the main flow. (Carvalho, 2010).

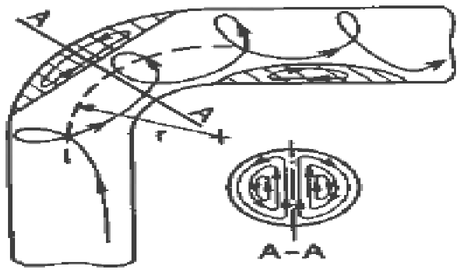

Figure 1 - Regions of separated flow in a hydraulic curve. (Carvalho, 2010).

The study of these regions of separated flow is difficult to carry out by hydraulic experiment, the use of computing tools that allow the visualization of the distribution of velocities within the working geometry is required. Among these tools highlights the computational fluid dynamics (CFD).

The CFD technology appears as one of the most viable ways to develop equipment related to any type of flow. According Gianelli (2010), computational fluid dynamics allows all types of behavior of a fluid are studied within a geometry in any virtual environment, which provides agility and lower costs in relation to development projects. Cipolla (2009) stated that numerical tool can be used to help bring the numerical study of the fundamental characteristics of flow experiments. These numerical simulations allow optimization projects at low cost without the need for large number of experiments for data acquisition.

The study of velocity fields in hydraulic devices by means of CFD techniques has been studied by several authors. Neto et al. (2008) conducted a numerical simulation of flow in a pipe, a valve-type focusing drawer through ANSYS CFX software. From the simulation the author was able to study the profile of the velocity vectors and velocity fields in the attachment. Both the results obtained via physical laboratory, as calculated from the theory and those obtained by numerical simulation were very close. Already Santos et al. (2013) conducted a study of velocity fields in a distribution pipe with flow in motion through vertical tubes and concluded that this provides a detailed analysis and visual quality of various characteristics of the flow.

Tu et al. (2008) also presents applications of CFD tools in various fields such as: aerial sciences, automotive engineering, biomedical engineering, chemical process, civil and environmental engineering, power generation, etc.

This paper aims to carry out the study of velocity fields and flow characteristics in a long-distance curves of the Laboratory of hydro Small Hydropower (LHPCH) of the Federal University of Itajubá (MG), using a model in virtual lab using the software of CFD.

2 Methodology

The fluid dynamic simulations were performed at the Center for Modeling and Simulation Environment and Water Resources (NUMMARH) of the Natural Resources Institute of the Federal University of Itajubá (MG) using CFD software.

The work was created geometry was created in Design Modeller application environment ANSYS Worbench being identical to curve that is located in the Laboratory of hydro Small Hydropower (LHPCH) of the Federal University of Itajubá (MG) presented in the figure 2. The diameter of the curve is equal to 200 [mm], and it consists of cast iron. The equivalent roughness adopted for this pipe was $\varepsilon = 0.26$ [mm]. A lot of straight pipe of 0.2 [m] was also built before and after the bend in order to view the properties of the flow in these regions:

Figure 2 - Hydraulic Curve studied.

The generation of the meshes took place within CFD software. These were built more refined way in curvature illustrated in figure 3 and more rustic in other regions:

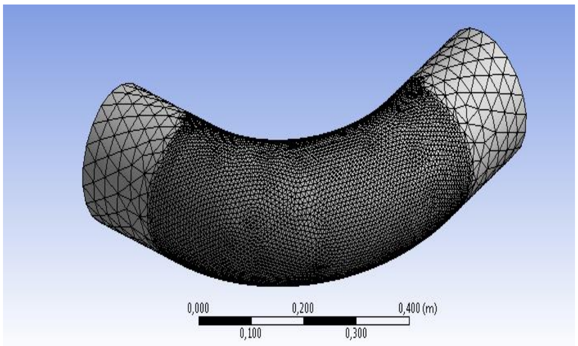

Figure 3 - Geometry and resulting mesh

The boundary conditions were activated in the setup application within the CFD software. The simulation scenario was defined with an inlet pressure Pe = 20 [kPa] and output speed of 1,591 [m / s] (flow rate Q = 50 [l / s]). For the simulations we adopted was that the flow isothermal (no temperature changes) with temperature at 22øC. Still in the CFD Setup application were selected features of the solver (solver). The minimum and maximum number of iterations chosen, were respectively 1 and 10000 The convergence criterion was less than 10-5 residue.

3 Results

The distributions of the velocity vectors of the flow lines and the curve obtained by computer simulation, are presented below (with flow direction from the top to the bottom) illustrated figure 4 e 5.

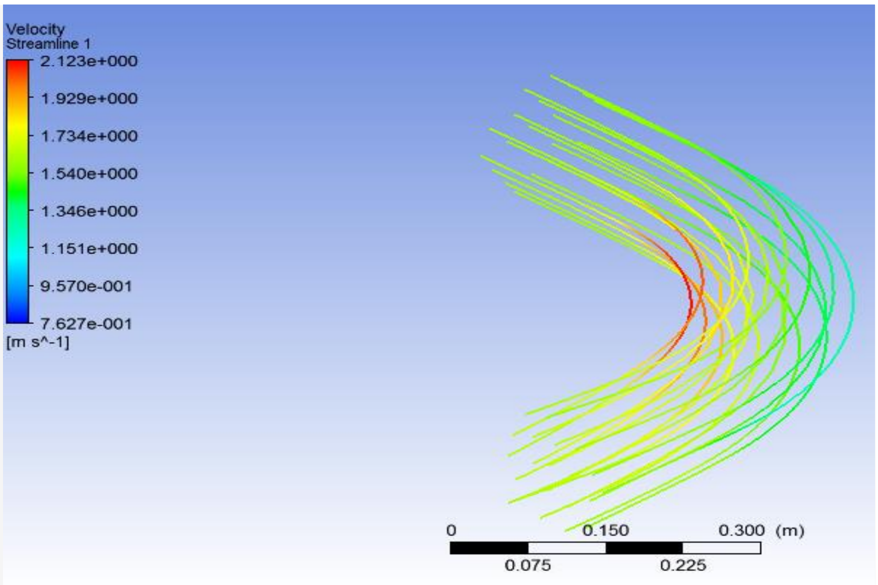

Figure 4 - Distribution of the flux lines curve.

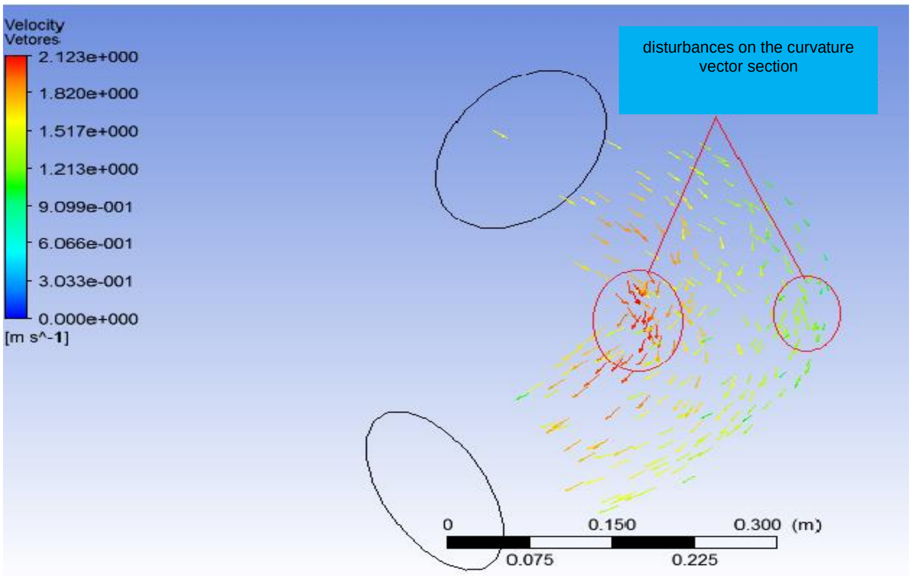

Figure 5 - Distribution of velocity vectors on the curve.

The analysis of the figures 4 and 5 allows us to observe that the convex part of the curve showed the higher rate while the hollow showed lower values, thus featuring a velocity gradient between the ends of the curve.

The figures 6 and 7 also show lines of disruption of flow and velocity vectors near the ends of the curve. These disturbances both intensity and direction of the flow lines and velocity vectors are responsible for the generation of turbulence in the flow, the fluid imposing a loss of localized charge and pursue the straight section of pipe after the bend until the moment that the flow reaches steady state again.

The Figure 6 and 7 shows the velocity distribution in a transverse plane located at the very beginning of the curve. The plane intersects a small portion of the straight section:

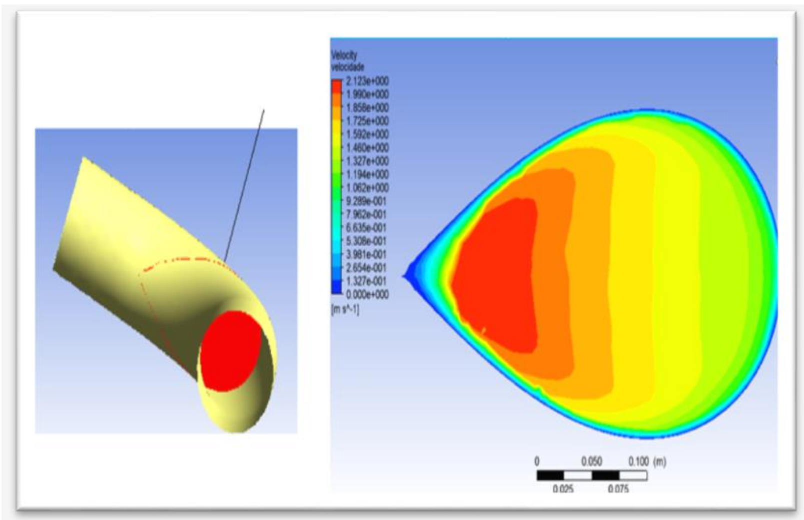

Figure 7 Distribution of speeds in a transverse plane.

For the analysis of Figure 7 shows the regions of different velocities which are formed in the cross section of the curve. Figure 8 below shows the velocity vectors in this section together with the input and output sections. It can be visualized very clearly the change of the flow change and how this change imposes disturbances on the velocity profile:

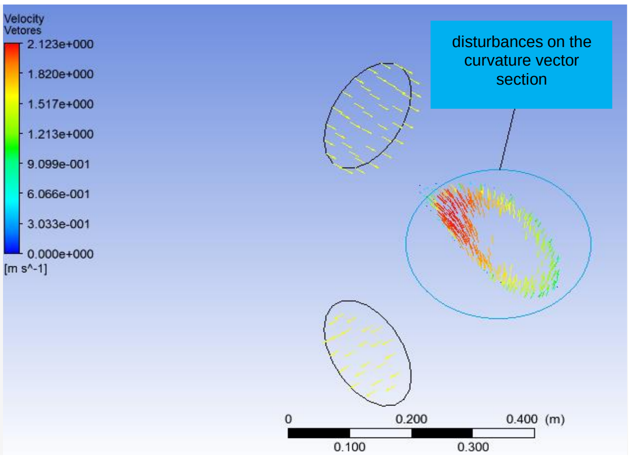

Figure 8 Disturbances on the velocity vectors in the bend section.

4 Conclusions and recommendations

We conclude that the velocity fields in flows are sensitive to changes in geometry and to be analyzed efficiently should use computational tools.

The results obtained in computer simulation with the help of CFD program, demonstrates that this provides a detailed view of various characteristics of the flow, such as disturbances of the velocity vectors, the velocity gradient between the ends of the curve and regions of different speeds runoff.

It is recommended for future work in the area to investigate how the velocity values at the ends of the curve vary under different conditions of pressure and flow.

5 - References

- CARVALHO, N.P.M. Losses and Load Dimensional Laminar Flow in Ducts with buckle 90^0. Faculty of Engineering, University of Porto, 2010 58p. Dissertation to obtain the title of Master in Mechanical Engineering.

CIPOLLA, E. Z. (2010). "Evaluation of the hydraulic behavior of a centrifugal pump using CFD tools." Natural Resources Institute, University of Itajubá MG, 100p. Thesis (monograph).

GIANELI, R. C. (2010). Computational modeling and simulation of Venturi meter with multiple pressure taps. Final Graduation Work of Hydroelectric Engineering. Federal University of Itajubá.

Neto, H. J, SILVA,F.G.B. Modelling and simulation of a valve hydraulic flow using the Hydro tool. Magazine Tecnologies. Fortaleza, v. 29, n. 2 p.224-232, 2008.

SANTOS, I. S. F, SILVA,F.G.B, BARROS, R.M., FILHO, G.L.T. Studies of velocity distribution with CFD techniques for branch piping water in water supply system. Accepted for publication PCH Notícias & SHP News, 2013.

PORTO, R, M. Basic Hydraulics, 4th Ed, 2006.

TU, J., et al, Computational Fluid Dynamics:. A Practical Approach. Ed. Elsevier, 2008,452p.