CFD ADJUSTED ONE-DIMENSIONAL JET PUMP MODEL

Glauber Cruz, glauber@ita.br
Edson Luis Zaparoli, zaparoli@ita.br
Claudia Regina de Andrade, claudia@ita.br
Instituto Tecnológico de Aeronáutica, Departamento de Engenharia Mecânica e Aeronáuitca - Praça Marechal Eduardo Gomes, 50 – Vila das Acácias – CEP: 12228-900 - São José dos Campos (SP)

Abstract. *This work deals with the numerical simulation of an isothermal fluid flow in an axisymmetric jet pump using a CFD tool. Considering an incompressible constant properties turbulent flow, the numerical solution was obtained applying a segregated approach with finite volume discretization of the governing equations. The one-dimensional loss coefficients were evaluated using the numerical results. The one-dimensional jet pump model results were compared with experimental results for suction flow rate, head ratios and efficiencies. This adjusted one-dimensional model is important to the preliminary design of the jet pump.*

Keywords: *jet pump, numerical solution, CFD, eject pump, efficiencies, performances*

1. INTRODUCTION

The jet pumps are devices appropriate to pump a fluid that can be liquid, gas or vapor or a two-phase mixture. Such devices are characterized by the exchange of the kinetic energy of the primary fluid with the secondary fluid in a mixture chamber. Due the simplicity in the structure, in the absence of the mobile parts and convenience of the maintenance, jet pumps have been used in many fields of the engineering to some many purposes. Jet pumps are used mainly in fields as: pumped fluid being or suctioned, dragged, chemical products being pumped and transport of great solid particles or until same foodstuffs (Kwon et al., 2002). A jet pump is geometrically simple since that it consists of four main components: a nozzle, chamber of suction, mixer throat and diffuser. The Computational Fluids Dynamics (CFD) tool was used to calculate the three-dimensional fluid flow (water) in the inside of a jet pump with geometric and performance data available in literature. The jet pumps works in the principle of the direction of the jet entering in the suction of flow. In general, jet pump conventional, a jet pump central, has a nozzle in the center where a primary fluid is injected and a suction part annular. Many researchers (Ueda, 1954; Sanger, 1970; Yano, 1990; Jo, 1996) have studied the performance of this type of jet pump. Other types of jet pump, jet pump annular, have one nozzle in the exterior tubing and suction in the center.

To see the effect of the form of the chamber of the mixture in the performance of the jet pump for various rates of flow of jet pump, jet pumps was tested by some of the authors (Oh et al., 1998), which were projected for the transport of fish. Diverse computational iterations are made, concluded and the results are argued to supply better given a detailed development of the flow jet pump annular.

A bigger attention has been directed, route to the optimization of geometry, performance of the cavitation, period of training of operation and the operation of the characteristics of low rate of area of the jet pumps. Analytical and empirical relations have been developed which accurately predicted both without cavitation of the performance of the flow bombs. (Sanger, 1970)

The jet pumps has been considered as a bomb of impulse auxiliary in cycle Rankine with the power of the electric space generating systems because of its simplicity, trust and good performance to low the positive suction in the head. Systems operating in conditions result in the petition where the flow bombs have low ratio of area in the exit of the nozzle for the area of the throat (area ratio, R). Decrease had little commercial interest in the development of low the ratio of area because of its characteristic of main ascension. (Sanger, 1969)

According to Yano et al. (1990), jet pumps to the water have low efficiencies comparative to the others types of the pumps as the pumps centrifugal machines. However, they are slightly used in systems of circulation of chemical plants, water bombardment, and draining in lines, because with the advent of the simple construction without some parts moving. Before more, types special there of bombs of flow to the water are which are convenient to carry liquid contend great solid particles.

This work deals with the numerical simulation of an isothermal fluid flow in an axisymmetric jet pump using a Computational Fluids Dynamics (CFD) tool. Considering an incompressible constant properties turbulent flow, the numerical solution was obtained applying a segregated approach with finite volume discretization of the governing equations, for in such a way the equations of conservation of the mass, second law of Newton, conservation of energy and for the model of turbulence k-ε, had been decided using the method of finite volumes with an algorithm segregated for the coupling of the fields of velocity and pressure. The one-dimensional loss coefficients were evaluated using the numerical results. The one-dimensional jet pump model results were compared with experimental results for suction flow rate, head ratios and efficiencies. This adjusted one-dimensional model is important to the preliminary design of the jet pump.

2. NUMERICAL MODELING

In this work the average turbulent flow of a viscous Newtonian fluid is considered, incompressible and in steady state. The instantaneous values for the pressure and velocity fields can be decomposed in one average component and floating parts. The governing equations are gotten of the Navier-Stokes equations. (Mompean, 1998)

2.1. Parameters for the project of Ejectors Bombs

The objective of this work was to simulate the flow in an ejector bomb that operates inside in the maximum efficiency of the specified restrictions: taxes of flow primary, secondary and pressures.

The hypotheses that are common to each analysis without cavitation are: (1) both the fluids, secondary and primary and, are incompressible. (2) the temperatures of fluids primary and secondary are equal. (3) ratio enters in the distance of the exit of the nozzle until the entrance of the throat and the diameter of the throat is zero ($s/d_t=0$). (4) the thickness of the wall of the nozzle is zero. (5) an additional assumption is used in the analysis, that the mixture of two fluids is complete in the exit of the throat.

The parameters of project of an ejector or the ejectors bombs are generally express in terms of the dimensional less relations that describe the geometry and the performance of the device. The five used main relations of the project are defined to follow.

i) Ratio between the length (l) and the diameter of the mixer chamber or throat (d_t):

The efficiency of a ejector bomb depends on the ratio between the length and the diameter of the throat. In this work such ratio was admitted as $l/d_t=3.54$cm. (Sanger, 1969)

$$\frac{l}{d_t} \tag{1}$$

ii) Ratio enters the spacing of the nozzle (s) and the diameter of the throat (d_t):

$$\frac{s}{d_t} \tag{2}$$

iii) Ratio enters the areas (R) of the exit of the nozzle (A_n) for the entrance of the throat (A_t):

$$R = \frac{A_n}{A_t} \tag{3}$$

An ejector bomb of ratio of small area (R=0.066), is characteristically of low pressure and raised ratio of secondary and primary flow. Inversely, an ejector bomb with ratio of great area (R=0.197), is high-pressure and the flow ratio is low. It is considered, in the practical one that the band of values of R is of 0.066 to 0.197.

iv) Ratio between fluid flows (M).

$$M = \frac{Q_2}{Q_1} \tag{4}$$

v) Ratio of manometric height (H):

$$N = \frac{(P_5 - P_2)}{(P_1 - P_5)} \tag{5}$$

When the fluids primary and secondary are of equal density, (N) can be express for the ratio of height of bombardment, express as:

$$N = \frac{(H_5 - H_2)}{(H_1 - H_5)} \tag{6}$$

vi) In such a way, the efficiency (η) of an ejector bomb can be express for:

$$\eta = \frac{Q_1(P_5 - P_2)}{Q_2(P_1 - P_5)} = M \times N \tag{7}$$

Remaining the too much fixed parameters, the efficiency of an ejector bomb is a function of the ratio between fluid outflows (M). In the assayed ejector bomb in Sanger (1969), the diffuser has an angle of opening (β) of 2.5º, and a ratio between areas of R=0.197. The maximum obtained experimental efficiency was of 37% for a spacing of the nozzle of and ratio of flow of M=1.6.

2.2. Mathematical equations

The average equations for the modeling of the internal and turbulent flow in ejectors bombs are presented here.

Second White (2002), the equation (8) represents the conservation of mass for flow in steady state of an incompressible fluid:

$$\frac{\partial \overline{U_i}}{\partial x_i} = 0 \tag{8}$$

In accordance with Okiishi et al. (1971), the second law of Newton or rocking of amount of movement is express for the equation (9).

$$\frac{\partial \overline{U_i}}{\partial_t} + \frac{\partial}{\partial x_j}(\overline{U_i}\,\overline{U_j}) = -\frac{1}{\rho}\frac{\partial \overline{P}}{\partial x_i} + \frac{\partial}{\partial x_j}(\nu \frac{\partial \overline{U_i}}{\partial x_j} - \overline{u_j u_i}) \tag{9}$$

Where υ and ρ are, respectively, kinetic viscosity and the density of the fluid. The amount $\overline{u_j u_i}$ is the Tensor of tensions of Reynolds.

In years 70, Hanjalic et al., (1972) was developed the standard version of the model k-ε. The form of the shaped equations was based on a phenomenological boarding of the turbulence and had been incorporated many of the previous ideas of the model of Prandtl-Kolmogorov. (Jones et al., 1972)

The linear model k-ε isotropic classic based on the approach of Boussinesq uses the following relation enters the components of the tensions of Reynolds and turbulent viscosity:

$$-\overline{u_i u_j} = 2\nu_t \overline{D}_{ij} - \frac{2}{3}k\delta_{ij} \tag{10}$$

Where D_{ij} it is the average rate of deformation:

$$\overline{D}_{ij} = (\frac{\partial \overline{U}_i}{\partial x_j} + \frac{\partial \overline{U}_j}{\partial x_i})/2 \tag{11}$$

The turbulent viscosity μ_t can be calculated by:

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon} \tag{12}$$

Where c_μ it is a constant and she is used to modify the values of μ_t next to the wall, where the effects of the diffusion molecular viscose are very important. The turbulent kinetic energy and the rate of dissipation are expressed in (Fluent, 1998) as being:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left(\left(\frac{\mu_t}{\sigma_k}+\mu\right)\frac{\partial k}{\partial x_i}\right)+G_k+G_b-\rho_\varepsilon-Y_m \tag{13}$$

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_i}\left(\left(\frac{\mu_t}{\sigma_\varepsilon}+\mu\right)\frac{\partial \varepsilon}{\partial x_i}\right)-\rho C_{\varepsilon 2}\frac{\varepsilon^2}{k+\sqrt{\nu\varepsilon}}+C_{\varepsilon 1}\frac{\varepsilon}{k}C_{\varepsilon 3}G_b \tag{14}$$

Where G_k it represents the generation of turbulent kinetic energy due to the average gradients of velocity; G_b it represents the generation of the turbulent kinetic energy to the push; Y_m it represents the contribution of the fluctuations of density in a turbulent compressible flow in the rate of dissipation of k; and σ_k and σ_ε are the turbulent Prandtl numbers for k and ε respectively;

2.3. Numerical Method

As for the numerical solution of the equations presented in the previous a commercial package was used (Fluent 6.3.17), in this computational method is described succinctly. With this it is intended to present the main steps that must be executed to get a numerical solution.

It is intended to get, through a numerical solution, values of dependent variable such as velocity and pressure in discrete points of the physical domain (mesh – see Fig. 3 and 4). The relation between noded values is determined by algebraic equations that are derived from the distinguishing equations in set with appropriate functions of interpolation for the dependent variable. The discretization equations then are decided by iterative methods. The method of discretization used in this work was the Method of Volumes Finite (MVF) for discretization of the governing equations.

The discretization of each one of the equations of amount of movement results in a system of algebraic equations for each component of the U_i velocity (u, v and w), that is, three systems of algebraic equations. With the discretization of the equation of the continuity and substitution of the U_i components, gotten of the equations of discredited amount of movement, an algebraic form for the equation of Poisson of the pressure is gotten. With the use of the discredited equation of Poisson one another system of algebraic equations is also gotten.

This set of 4 (four) systems of algebraic equations is not linear, therefore the terms of each one of the matrices of coefficients depend on the dependent variables of the proper system and the too much systems. In the used iterative method for solution (segregated), each one of the systems is decided sequentially and repeatly. For example, the system for the component **u** of the velocity is decided using the values of the excessively changeable dependents gotten in the previous iteration. After that, the system of equations for **v** and for all is decided successively/repeatly the other dependent variable. It must be standed out that each one of the systems of algebraic equations also is decided iteratively.

For the use of this procedure and numerical solution of the flow in a domain the representation of this for a mesh is necessary. For in such a way, the domain (in this case - region where the flow occurs) is divided in small elements (volumes of control). In the "center" of each one of these volumes of control it is a point of the mesh where the dependent variable are calculated the values of all. The domain (infinite number of points) starts to have a discrete representation: finite number of points of one threshes. The main characteristics of each point of the mesh are: coordinated and relation of neighborhood with other points.

In the process of transformation of the distinguishing equations in algebraic equations, projects are used that have some similarity with finite differences. The derivatives are represented of form approached for "finite differences" and the error depends on in the distance enters the neighboring points of the mesh, that is, it depends on the refinement of the mesh.

When the distances between the neighboring points of the mesh are lesser (number of points to represent a very bigger domain) lesser are the errors of the numerical solution. The number of points in one threshes uniform in a domain increases quickly with the reduction of in the distance between the points. With this it also grows in a not linear scale the computational work to decide each one of the systems of algebraic equations and the capacity of storage of the computer for the six variable for point of the mesh (**u, v, w, p, k** and **ε**) - memory and disk

The quality of the gotten numerical results depends on a solution of commitment between number of points of the mesh, residues of the solution of the systems of algebraic equations, available time of solution and computational resources.

To establish the quality of a numerical solution, the influence of the mesh in the results must as a first step is analyzed, that is, to verify that the resultant error of the approach of the derivatives for “finite volumes” is not dominant. This study it has as objective to show that the gotten numerical results its does not depend of the mesh.

One the same simulated fluid in all the entrances, water in the liquid state, remaining its constant temperatures and properties.

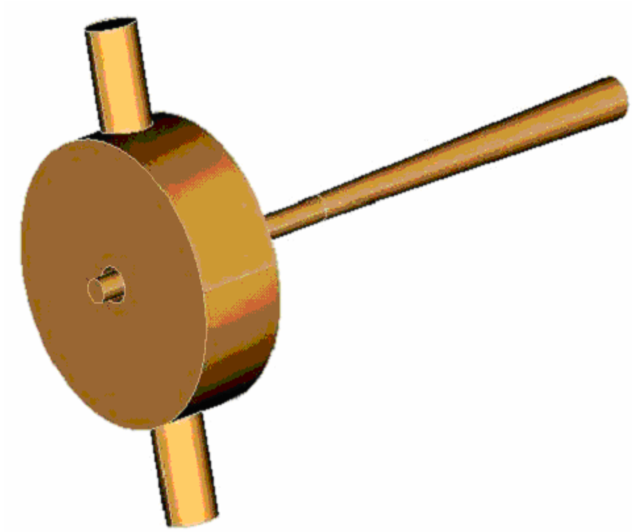


Figure 1. Sight frontal of geometry 3-D

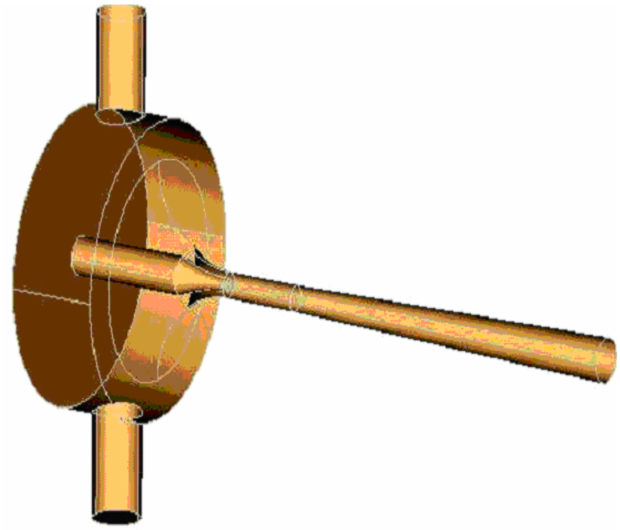


Figure 2. Lateral sight and internal of the geometry 3-D

One meshes tetrahedrycal that was used to represent the domain of the draining is presented in Fig. 3. This mesh to re-cover all the regions of the domain contained 1.086.175 cells, with refinement in the regions most critical, where intense variations (high gradients) of velocity and pressure occur such as: in the peak of the ejector, throat of mixture and diffuser (see Fig. 4).

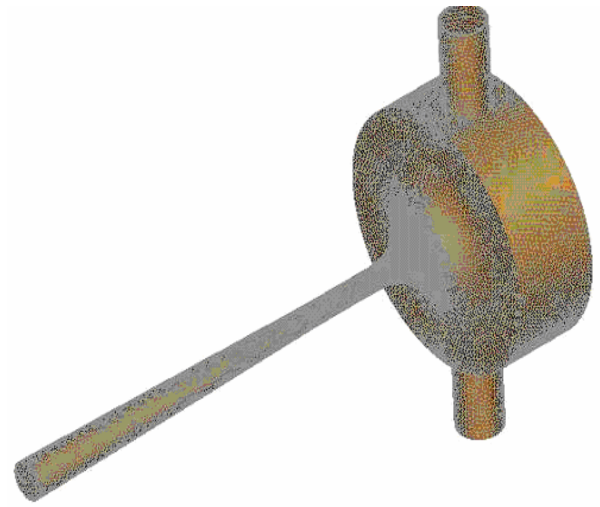


Figure 3. Mesh throughout all geometry

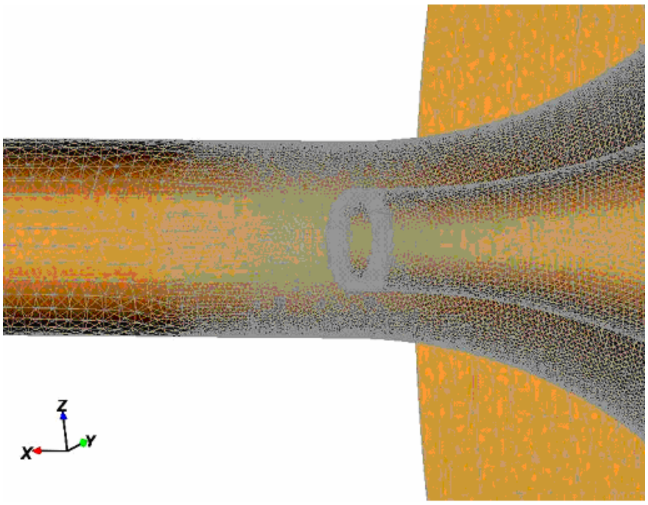


Figure 4. Zoom of the mesh at the beginning of the throat and end of the injector peak

3. RESULTS AND COMMENTS

Diverse computational simulations had been carried through for the ratio of flow of secondary and primary draining, with the purpose experimentally to compare the results simulated with the gotten ones.

One of the first parameters to be investigated in this work, was the efficiency of the ejector bombs, in accordance with the equations (1 to the 7), as a function of the ratio enters the fluid outflows (M) and ratio of manometric height (N).

In the Fig. 5 it is presented variation of the efficiency of the jet pump in function of the ratio of flow. Bigger differences occur between the numerical results and of the one-dimensional theory, mainly for the central region of the band of ratio of analyzed flow. The comparison of the results for the ratio of manometric height is presented in Fig. 6.

Inside of the observed one in figures 12 and 13, one perceived that the numerical values had presented a good agreement with the experimental results (Sanger, 1969) and with the one-dimensional theory. Valley to remember that the one-dimensional theory, takes in consideration the losses for attrition throughout the ejector pump and also the coefficients had been adjusted experimentally by (Sanger, 1967), for the presented experimental results in the Fig. 5 and 6.

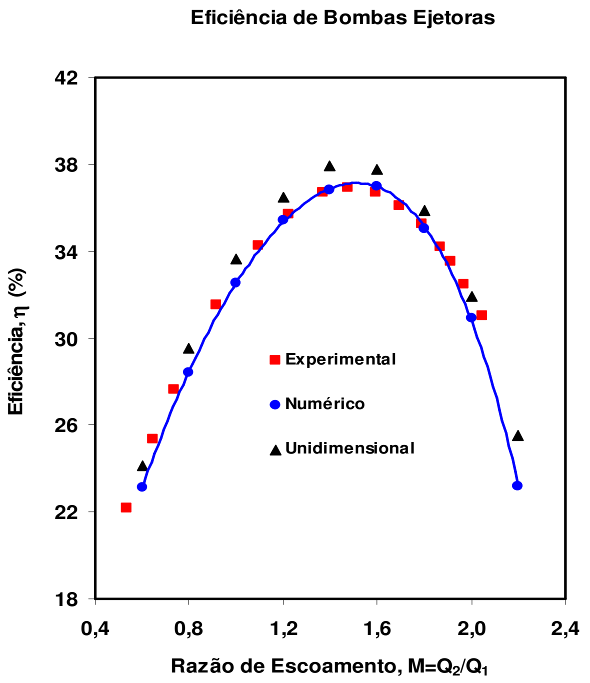


Figure 5. Efficiency (η) of the ejector bomb in function of the Ratio of Flow (M)

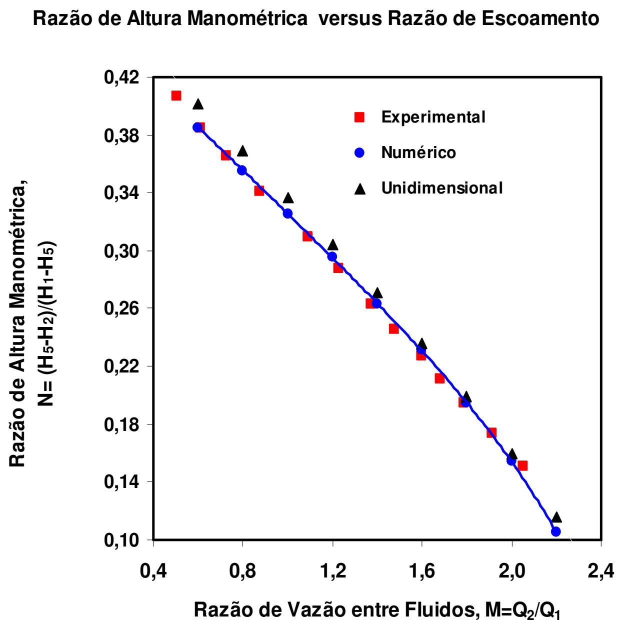
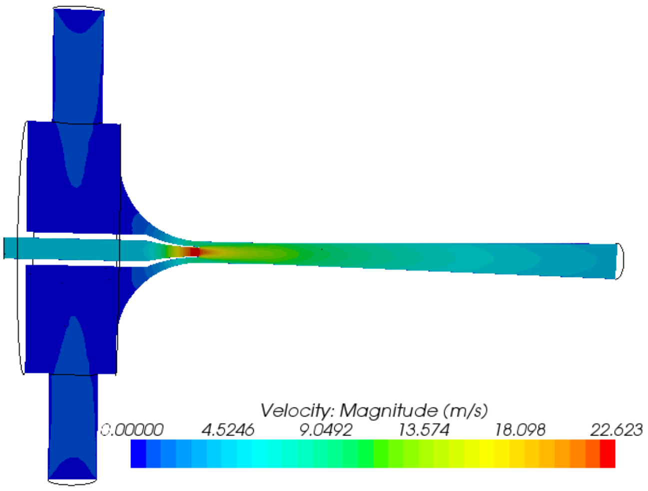


Figure 6. Ratio of Manometric Height (N) versus Ratio Flow between Fluids (M)

The values of the magnitude of velocity in the ejector bomb are illustrated in Fig. 7. One notices that the primary fluid is sped up in the nipple (where the biggest values of the velocity occur) and produces a nucleus of high velocity in the mixer throat. A detail of the draining in chamber of mixture is presented in Fig. 8.



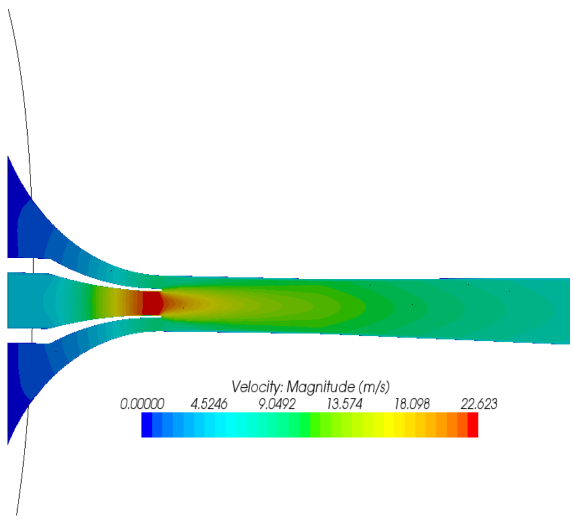


Figure 7. Longitudinal and central cut for magnitude of velocity throughout the ejector bomb

Figure 8. Detail of the field of velocity in the mixture chamber

It is verified that the main jet (central) if mixture with the secondary jet dragging molecules of this. This provokes a suction that induces the draining of the secondary fluid. The Fig. 9 shows the distribution of static pressure in one longitudinal and central cut of the jet pump. The high pressure of the main draining is converted into velocity during the passage for the nozzle. Throughout the extension of the chamber of mixture and the diffuser, the kinetic energy of the main flow is changed into energy of pressure (Fig. 9 and 10), first, for the process of mixture in the stretch of straight pipe and later, for the mixture and increase of the area of flow in the diffuser.

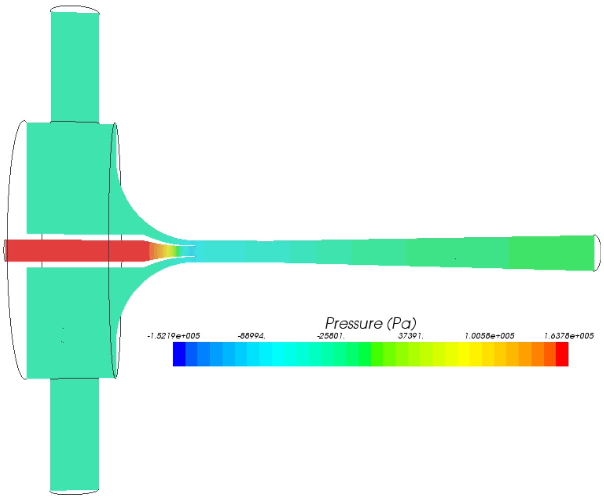


Figure 9. Distribution of static pressure in one longitudinal and central cut of the jet pump

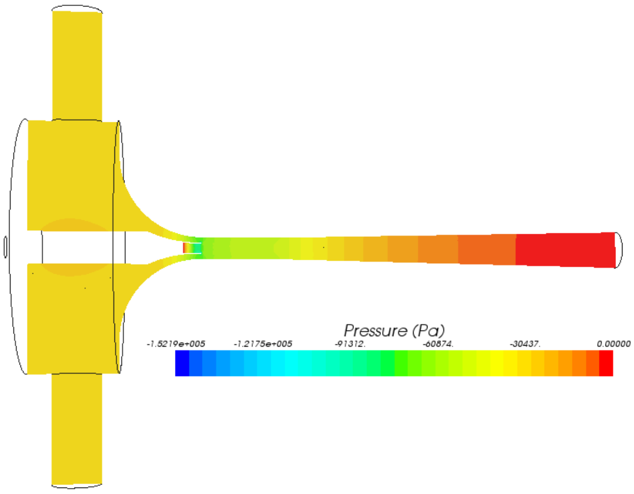


Figure 10. Part of the domain with negative static pressure in the interior of the jet pump

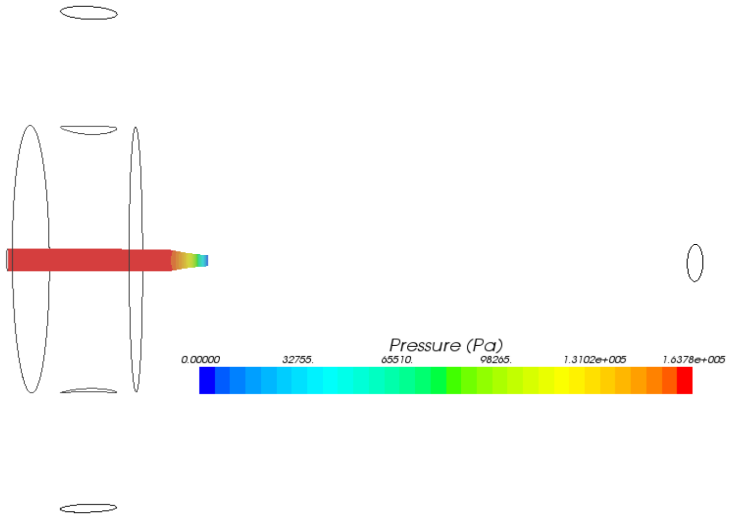


Figure 11 - Part of the domain with positive static pressure in the inside of the jet bomb

It is verified for Fig. 10 that since a transversal section next to the exit to the peak ejector, all the ejector bomb operates with negative pressures (having as reference a null value for the pressure in the exit of the diffuser). In Fig. 11 the pressure in the secondary jet is noticed that positive pressures only occur in the primary jet and that negative pressures in the secondary jet represent the suction, therefore is minor who the atmospheric pressure (imposed as condition of contour in the exit of the diffuser).

The flow with geometry was simulated equivalent (computational mesh with about 1 million of cells) to presented in (Sanger, 1969) for ratio between area of the nozzle/throat equal 0.197 and the numerical results had in such a way presented good agreement with the experimental data as with the results of the one-dimensional theory. Valley to remember, that this on the basis of had its coefficients of loss for attrition adjusted to the experiments carried through by Sanger (1969).

They had been reached maximum of efficiency of the bomb ≈ 37% for ratio between the flows secondary (Q_2) and primary (Q_1) equal at 1.6. An almost-parabolic behavior for the efficiency of the bomb in function of the relation Q_2/Q_1 was verified. Already for the relation of manometric heights, the dependence is practically linear in function of the ratio Q_2/Q_1.

Beyond the parameters for which it had given experimental for comparison, the computational tool allowed to analyze the behavior of other physical size as the distribution of static pressure and the field of velocity in the elements of ejector bomb (nozzle, mixer throat and diffuser). These results had allowed evaluating the regions of positive or negative pressures, the quality of mixture of the flows primary and secondary and the effect of these parameters in the efficiency reached for the ejector bomb. On the basis of these results, new geometric configurations also can be simulated, without the necessity of the construction of the archetype and reducing the costs and time for attainment of results.

4. ACKNOWLEDGEMENTS

The authors would like to acknowledge to the Laboratory of Flow and Heat Transfer Simulation (GSET/ITA) and the CAPES by the financial support.

5. REFERENCES

Fluent Inc., 1998, "Fluent 5 User's guide". Lebanon, Vol. 2.
Hanjalic, K.; Launder, B. E., 1972, "A Reynolds stress model of turbulence and its application to thin shear flows", Journal of Fluids Mechanics, Vol. 52, pp. 609-638.
Jo, J. G., 1996, "A numerical analysis on flow in the jet pumps", Spring Congress on Korean Society of Computational Fluids Engineering, pp. 99-104.
Jones, W. P.; Launder, B. E., 1972, "The prediction of laminarization with a two-equation model", International Journal Heat Mass Transfer, Vol. 15, pp. 301-313.
Kwon, O. B. et al., 2002, "Two-dimensional numerical simulations on the performance of annular jet pumps", Journal of Visualization, Vol. 5, No. 1, pp. 21-28.
Mompean, G., 1998, "Numerical simulation of turbulent flow near a right-angle corner using the speziale non-linear model with RNG K-ε equations", Computers and Fluids, Vol. 27, No. 7, pp. 847-859.
Oh, B. H.; Kwon, O. B., 1998, "A study on the characteristics of annular jet pump for various nozzle shapes", Bulletin of Korean Society of Fisheries Technology, Vol.34, No. 4, pp. 442-449.
Okiishi, Theodore H. et al., 1977, "Fundamentos da mecânica dos fluidos**"**, 2ª ed. São Paulo, Brazil: Edgard Blücher, Vol. 1.
Sanger, N. L., 1967, "Noncavitating performance of two low-area-ratio water jet pumps having throat lengths of 7.25 diameters", Washington, DC: NASA, (NASA TN D-4445).
Sanger, N. L., 1969, "Noncavitating and cavitating performance of several low area ratio water jet having throat lengths of 3.54 diameters", Washington, DC: NASA, (NASA TN D-5095).
Sanger, N. L., 1970, "An experimental investigation of several low-area-ratio water jet pumps", Transactions of the ASME, Journal of Basic Engineering, Vol. 92, No. 1, pp. 11-20.
Ueda, T., 1954, "Study on the water jet pump", Transactions of the JSME, Vol. 20, pp. 25, 1954. Coimbra, A.L., 1978, "Lessons of Continuum Mechanics", Ed. Edgard Blücher, São Paulo, Brazil, 428 pp.
White, Frank M., 2002, "Fluids Mechanical", 4ª ed. Rio de Janeiro, Brazil.
Yano, H. et al., 1990, "Performance of a new type of the water pump", Transactions of the ASME, Journal Engineering for Industry, Vol. 112, pp. 172-174.

6. RESPONSIBILITY NOTICE