INVESTIGATION INTO THE DYNAMICS OF THE INTERNAL FLOW IN A SWIRL ATOMIZER BY CFD SIMULATION USING ANSYS CFX

Roman Savonov, roman@ita.br
José Atílio Fidel Rocco Fritz, friz@ita.br
Koshun Iha, koshun@ita.br
Instituto Tecnológico de Aeronáutica, Praça Marechal Eduardo Gomes, 50, Vila das Acácias,
CEP 12.228-900, São José dos Campos, SP, Brasil
José Nivaldo Hinckel, hinckel@dem.inpe.br
Av. dos Astronautas, 1.758, Jd. Granja - CEP: 12227-010, São José dos Campos, SP, Brasil

Abstract. *This work presents the simulation of the internal flow in a swirl atomizer. The geometry of the atomizer is calculated by analytical equations used in engineering. The numerical simulation of the two-phase flow is performed by using two equations k-ε turbulence model. The fluids are presented as two-fluid homogeneous model. The interface between two phases is calculated by free surface model. The distribution fields of the axial and tangential velocities, pressures and air core are obtained. The aim of this work is to compare the results obtained by numerical simulation with ones obtained analytically. Also, to study the internal fluids flow inside the atomizer.*

Keywords*: Swirl atomizer; CFD simulation; ANSYS CFX*

1. INTRODUCTION

Swirl atomizers are widely used in rocket propulsion systems. They present a good quality of the atomization with relatively small length of the liquid spray in comparison with jet atomizers, and more simple design then others. Applications of swirl atomizers in rocket motors permit reduction of dimensions of the combustion chamber and, consequently, weight of the engine. Due of these properties, swirl atomizers were intensely investigated from 1940 up to present time.

The first approximate analytical solution of the ideal fluid flow inside the injector was present by Abramovitch (1944), six decades ago. This solution admits calculation of the mail characteristics of the injector and it`s geometry. Later, several researchers investigated flow processes inside the swirl atomizer with the aim to improve on the Abromovitch solution. Kliachko (1962) proposed the calculation of the coefficient of discharge using the equivalent characteristic instead the geometrical characteristic *A*. The equivalent characteristic includes the effects of friction losses inside the injector. All calculations in this method is based on calculus by Abramovitch theory. Kliachko considered that all hydraulic losses are due the friction. S. M. Double and E. M. Halton (1947) applied the theory of the cyclone to calculate the flow inside the atomizer, but this theory is based on the supposition that rotation of liquid is calculated by Eq. (1)

$$V_{\varphi}^{2} r = const \tag{1}$$

There are many publications about the investigation of the swirl atomizer, but all of them used the same process to obtain the parameters of injector. First is calculated the size of the air core, and then the mass flow rate through the atomizer is obtained. None of these works and proposed methods have any advantage over Abramovitch's theory. Therefore, analytical calculations of injector's parameters in this paper are based on the Abramovitch theory.

The computational power available to developers increased very much during the past decade. It stimulated the application of more sophisticated numerical methods and solution to investigate the flow dynamics of the atomizer.

Ales Alajbegovic *et al.* (2001) presented a three-phase flow simulation of the flow in the high-pressure swirl injector, with application in direct gasoline injection engines. The model was based on the multiphase extension of the two-fluid model. The authors obtained most important characteristics of the flow. A thin conical sheet with the air core was predicted. Cavitation, which occurs in the pressure depression located in the air core, was predicted. The analysis presented very promising approach for the simulation of flows in DGI injectors. Shanwu W. *et al*. (2007) presented an investigation on confined swirling flows in an operational gas-turbine injector. The study was performed by means of large-eddy simulation. The co- and counter-rotation configurations were considered, and the effects of swirl direction on flow characteristics were examined. Good agreement was obtained between the measured and calculated mean velocity fields and turbulence properties. This work demonstrated the feasibility of using LES to study complex flow fields. Hansen K. G. (2001) presented detailed investigation of the two-phase internal flow in a large-scale pressure-swirl atomizer. The study was performed by experimental measurement and CFD modeling using three approaches: volume of fluid model (VOF) with assumption a laminar flow, VOF using LES turbulence modeling, and two-fluid Euler/Euler method using a laminar flow assumption. All simulations presented a similar result and produced an air-core that matched those observed in the experiments. The tangential and axial velocity profiles in the conical swirl chamber and static wall pressure obtained by simulation demonstrated a good agreement with measured data. For the o flow rates in both cases, the VOF and Two-Fluid approaches assuming laminar flow appear to give the best agreement.

2. ANALISYS OF THE SWIRL ATOMIZER

The aim of this work is to investigate the internal flow in the injector. The internal flow characteristics, such as dimensions of the air core, distribution of the pressure inside the atomizer, field of tangential and axial velocities, take a key role in the formation of the liquid cone and atomization process. Therefore, it is very important to have full information about the behavior of a fluid inside the injector to predict more completely the processes, which occur after the fluid exits the atomizer, such as formation of the liquid cone, pulverization angel, thickness of the liquid sheet, atomization and others. The first phase of this work is to calculate the injector geometry.

2.1. Abramovitch's theory. Calculation of the dimensions of the atomizer.

In Abramovitch's theory, the friction losses in the injector are ignored, and the angular momentum in the swirl chamber is constant Eq. (2).

$$V_e R = V_t r \tag{2}$$

where V_e is the velocity of the liquid in the entrance of the atomizer;
V_t is the tangential velocity of the fluid;
R is the radius of the swirl chamber;
r is the radial coordinate of the liquid particle.

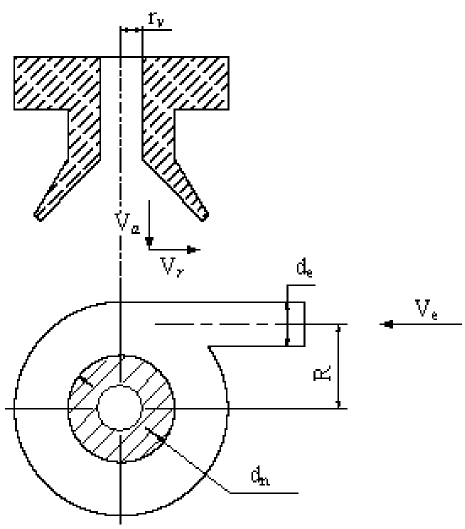


Figure 1. Swirl atomizer.

The liquid is considered to be inviscid. The pressure in any point of the injector is prescribed by Bernoulli equation. The radial velocity on the interface between the liquid and gas phase is considerate to be zero. The determination of the size of air core is based on the maximum consumption principle. Full description of the Abramovitch theory is discussed in Abramovitch (1944) or Vasiliev *et al*. (1993). Here are presented the equations to calculate the mail characteristics of the swirl atomizer, such as φ – coefficient of the nozzle filing Eq. (3), *A, B* – geometrical characteristics Eqs.(4, 5), μ - discharge coefficient.

$$\varphi = 1 - \frac{r_v}{r_n} \tag{3}$$

where r_v and r_n are radii of air core and nozzle respectively.

$$A = \frac{R r_n}{n r_e^2} \tag{4}$$

where n is number of the tangential orifice;
r_e is the radius of tangential orifice.

$$B = \frac{R}{r_e} \qquad (5)$$

The relation between φ, μ, A and *tan(α)* is prescribed by Abramovitch (1944) and Vasiliev *et al.*(1993) as;

$$A = \frac{1-\varphi}{\sqrt{\frac{\varphi^3}{2}}} \qquad (6)$$

The Eq. (4) is used to estimate the error of the theoretical calculus with experimental data, and Eq. (6) in initial calculation of the injector's geometry.

$$\mu = \sqrt{\frac{\varphi^3}{2-\varphi}} \qquad (7)$$

$$\tan\alpha = \frac{(1-\varphi)\sqrt{8}}{(1+\sqrt{1-\varphi})\sqrt{\varphi}} \qquad (8)$$

where α is a pulverization angle.

The Abramovitch theory is applied to an inviscid fluid. It can be applied to viscous fluid too. According to Vasiliev *et al.*(1993) if

$$\frac{B^2}{n} - A \leq 6.2 \qquad (9)$$

then friction losses can be considerate equal zero and Abramovitch theory give accurate results. The mass flow rate of liquid through the atomizer is

$$\dot{m} = \mu F \sqrt{2\rho\Delta P} \qquad (10)$$

where F is area of the injector's nozzle;
ρ is density of the liquid;
ΔP pressure difference;

Using equations above the dimensions and geometrical characteristics of the injector for nominal regime of water are calculated, for: ΔP =10atm, consumption 50 g/s, they are presented in Tab.(1).

Table 1. Dimensions and geometrical characteristics of atomizer.

Enter data				Parameters of the atomizer							
Pressure, atm	Consum-ption, g/s	Density, kg/m^3	Number of orifice	φ	μ	A	B	α,°	D, mm	d_e, mm	d_n, mm
10	50	998	2	0.345	0.158	4.569	3.658	105	4.39	1.2	0.3

To confirm and proof the precision of the calculus based on Abramovitch theory, the pressure was varied from 10 to 1 atm, and were calculated analytically the consumption, values of the axial and tangential velocities on the nozzle exit at itch pressure level for the same atomizer, and, than were compared with result obtained by numerical simulation using ANSYS CFX. To calculate these values were used equations presented by Vasiliev *et al*. (1993) or Khavkin (2004).

$$V_{eq} = \mu \sqrt{\frac{2\Delta P}{\rho}} \tag{11}$$

where V_{eq} is the equivalent velocity

$$V_a = \frac{V_{eq}}{\varphi} \tag{12}$$

where V_a is axial velocity in the exit of the nozzle.

$$V_t = V_{eq} \frac{R r_n^2}{r r_e^2} \tag{13}$$

$$\dot{m} = V_{eq} r_n^2 \pi \rho \tag{14}$$

The results of the calculations are presented in Tab.(2).

Table 2. Results of the analytical calculation.

ΔP, atm	m, kg/s	V_a, m/s	V_t, m/s	V_{eq}, m/s
10	0.049992467	20.57623	34,72147	7.100855
9	0.047427019	19.52032	32,93968	6.736463
8	0.044714622	18.40394	31,05583	6.351198
7	0.041826699	17.21531	29,05007	5.941002
6	0.038723999	15.93828	26,89514	5.500299
5	0.035350013	14.54959	24,55179	5.021063
4	0.031618013	13.01355	21,95979	4.490975
3	0.027382002	11.27006	19,01773	3.889299
2	0.022357311	9.201968	15,52792	3.175599
1	0.015809006	6.506774	10,97989	2.245488

2.2 Numerical simulation.

2.2.1 Mesh generation.

There are two types of grid generation: structured and unstructured. The geometry of the atomizer is not very complex, and it was possible to create the structured grid. In the beginning, a simplification of the geometry was assumed. The geometry of the entrance channels was substituted by orifice formed by intersection of these channels with the cylindrical part of vortex chamber, as shown at Fig. 2 (red region).

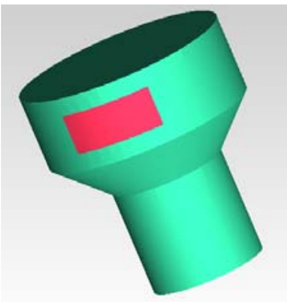


Figure 2. Atomizer`s geometry.

The multiblocks technique was used to generate the grid. The physical domain was divided by 69 blocks as shown in Fig. 3.

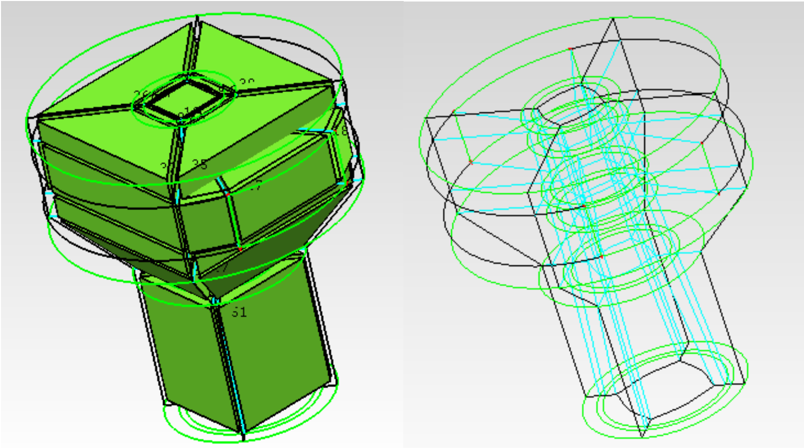


Figure 3. Block structure for grid generation.

The O'Grid function was used to obtain a good grid quality on the cylindrical regions. It is available in ICEM CFD software. Initially, homogeneous regular mesh without refinement regions was generated. The numerical simulation with this mesh did not present good results. The air core was formed only in the nozzle region, and velocity field near the walls had poor quality. But, this first analysis helped to determine the regions of higher gradient values. The block structure was adopted. The mesh was refined near the walls and on the interface between two phases water – air. Figure (4) shows the final mesh, which was used in numerical simulation.

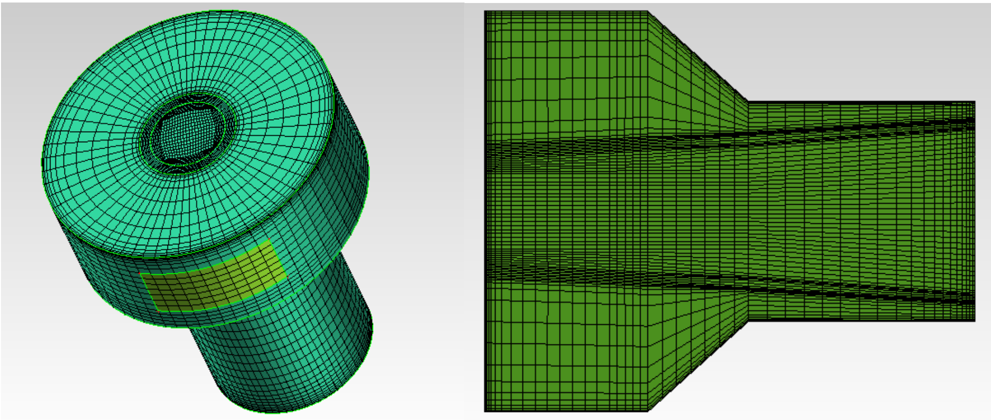


Figure 4. Mesh refinement.

22.2.2 Numerical model.

ANSYS CFX software has two ways simulate the multiphase flow, an ***Eulerian–Eulerian*** multiphase model and a ***Lagrangian Particle Tracking*** multiphase model. The two fluids - water and air, were considered to be a continuous fluids and ***Eulerian–Eulerian*** homogeneous multiphase model was used to simulate this flow. The free surface model was applied to calculate the interface between two fluids.

Reynolds number for the flow inside the atomizer can be calculated using Eq. (15), Vasiliev *et al*. (1993).

$$\mathrm{Re} = \frac{4\dot{m}}{(\pi \rho \nu \sqrt{nd})} = 4215 \tag{15}$$

The flow is turbulent for this value of Reynolds number. Two equations k-ε turbulence model was used in simulation.

The boundary conditions are: inlet – enter orifices, wall – atomizer`s walls, opening – the exit of nozzle.

At inlet boundaries total pressure from 1 up to 10 atm is applied. The turbulence kinetic energy and turbulence eddy dissipation can be calculated as:

$$k_{inl} = \frac{3}{2}(T_i\sqrt{u^2 + v^2 + w^2}) \tag{16}$$

$$\varepsilon = \frac{k^{\frac{3}{2}}}{0.3l} \tag{17}$$

where u, v, w – velocity component;
l – hydraulic diameter;
T_i – turbulence intensity.

where u, v, w – velocity component;
l – hydraulic diameter;
T_i – turbulence intensity.

At the opening boundary condition was applied a static pressure, and turbulence option is zero gradient.

Initial condition is static pressure equal to ambient pressure, water volume fraction is zero and air volume fraction is 1

2.3 Results.

The simulation is done for ten values (1 – 10 atm.) of the total pressure at entrance of the swirl atomizer. Figure 5 shows the air core formation corresponding to nominal pressure 10 atm.

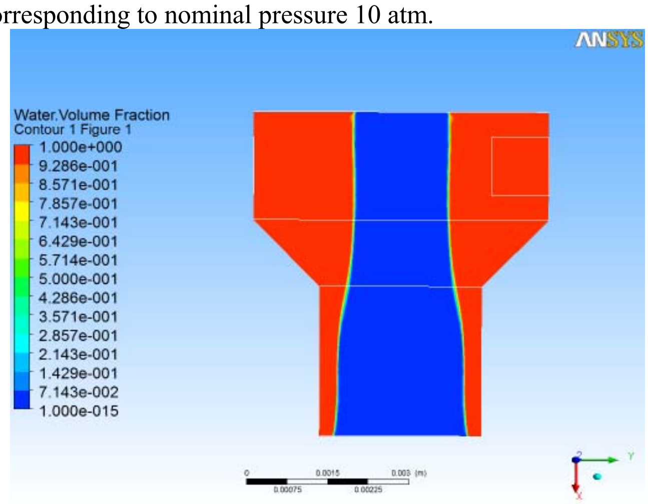
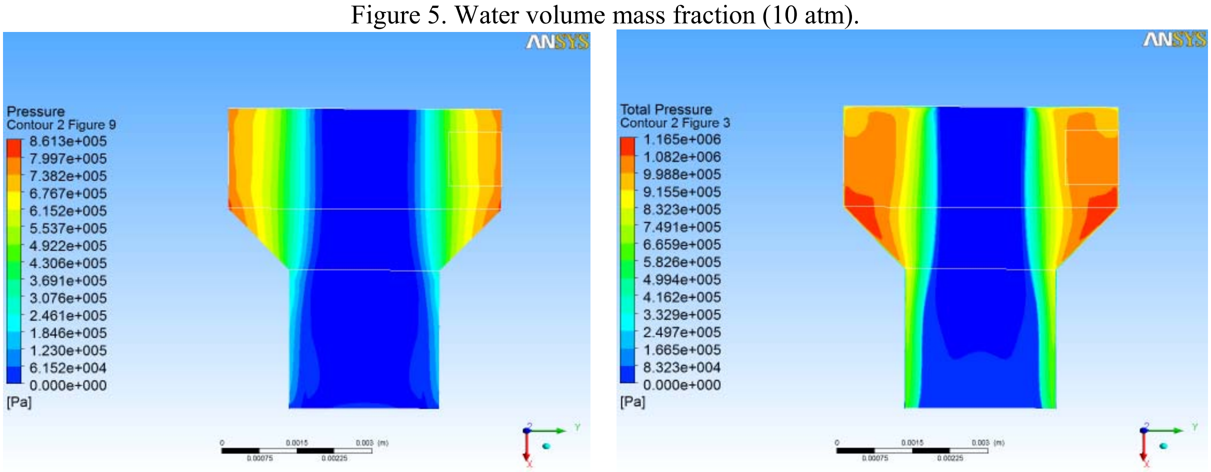


Figure 5. Water volume mass fraction (10 atm).



a) b)

Figure 6. Distribution of static a), and total b) pressures.

Figure (6) shows the distribution of the total and static pressures. It can be observed, that total pressure have higher values due the contribution of the dynamic component.

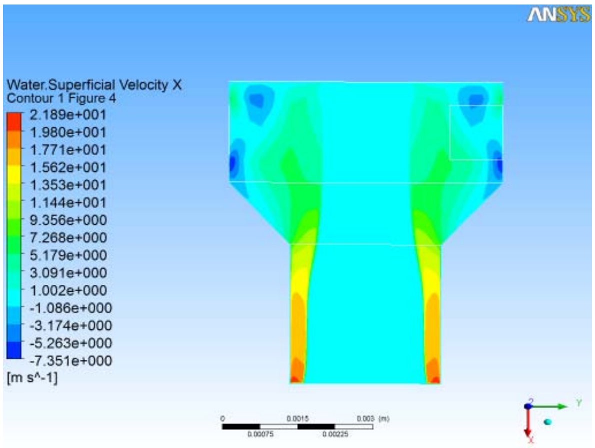


Figure 7. Water axial velocity.

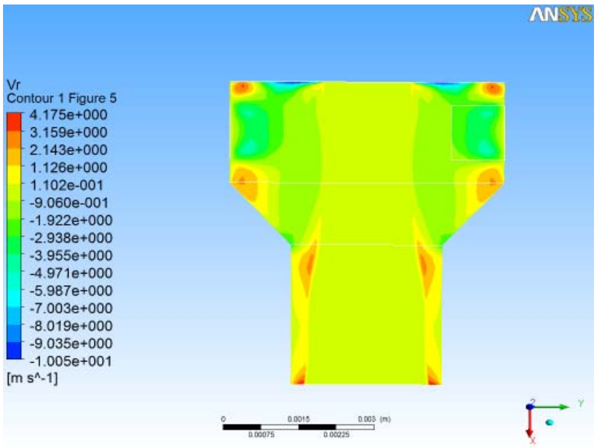


Figure 8. Water radial velocity.

Figure 7 presents the distribution of the water axial velocity inside the atomizer. Axial velocity has low values on the top of injector in the swirl chamber. In this region the axial velocity increases with approximation to the air core. With passage to the nozzle the axial velocity has greater values and reaches the maxim value on the exit of the nozzle.

Figure 8 shows the distribution of the radial velocity. Radial velocity has higher values on the two region of the nozzle – on the top part of the nozzle and on exit, due the increase of the radius of the air core. It can be observed comparing Fig. 8 and Fig. 5.

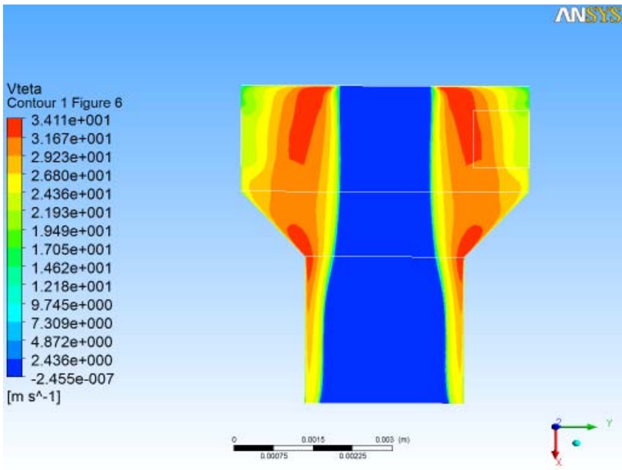


Figure 9. Water tangential velocity.

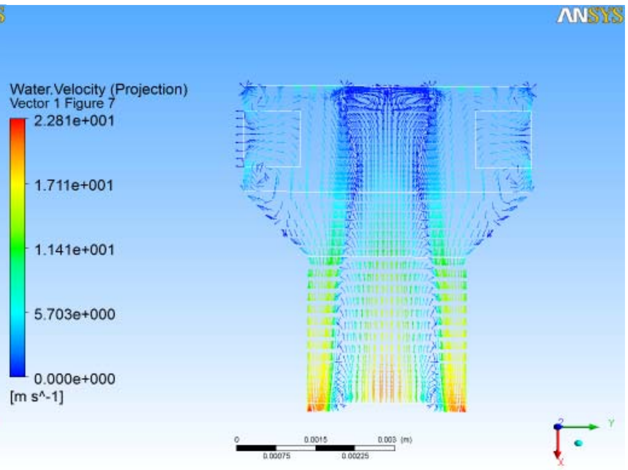


Figure 10. Vector plot of the water velocity.

The distribution of the tangential velocity is presented on the Fig. 9. The tangential velocity has greater values on the swirl chamber because the angular momentum in this part of the atomizer is greater then in the nozzle. It contradicts Abramovitch's theory, which is based on the assumption that the angular momentum is constant, and tangential velocities on the same radial location are equal, Eq. 2. But it has clear physical explication. The Abramovitch's theory considers the fluid to be inviscid, but the simulation uses a viscous fluid – water. The tangential velocity in the nozzle is smaller because the momentum in the nozzle decreases due of the friction losses. The behavior of the tangential velocity on the same section plane ZY is in accordance with the conservation of the angular momentum, Eq. 2. It increases when the radius reduces. Only in the region of the interface with the air core it decreases due of the interaction between water and air.

Analyzing Fig. 10 some recirculation zones can be observed. The recirculation zones are located on the swirl chamber above and below the entrance orifices. The fluid flow has hydraulic losses in these regions; therefore it is good to reduce the height of the swirl chamber up to the height of the tangential orifice. Others recirculation zones are concentrated on the interface of two fluid due the opposed flow direction of the air and water. The recirculation zone observed at the bottom of the swirl chamber. It appears because the air enters in the nozzle and flows off the top of injector. On top it changes the flow direction and flows off together water to the exit of the nozzle. The air core on the top has greater radius due of these recirculation zones.

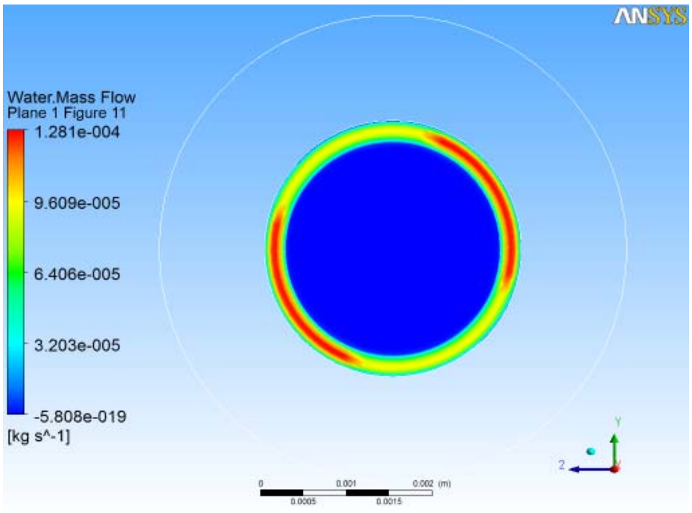


Figure 11. Water mass flow on the exit of the nozzle.

Water mass flow in the transversal section located on the exit of the nozzle is presented in Fig. 11. Two zones of the maximum mass flow can be observed here. This no-uniform distribution of the mass flow is due of the number of the tangential entrance of the injector. In this case the atomizer has two orifices of the fluid entrance, consequently, its form two zones of maxim mass flow. According to Vasiliev *et al*. (1993) no-uniformity reduces if the number of entrance orifices increase and the distribution of the mass flow is nearly uniform if number of the orifices is six. In case of the six orifices the zones of the maximum mass flow are closed together and the distribution becomes uniform.

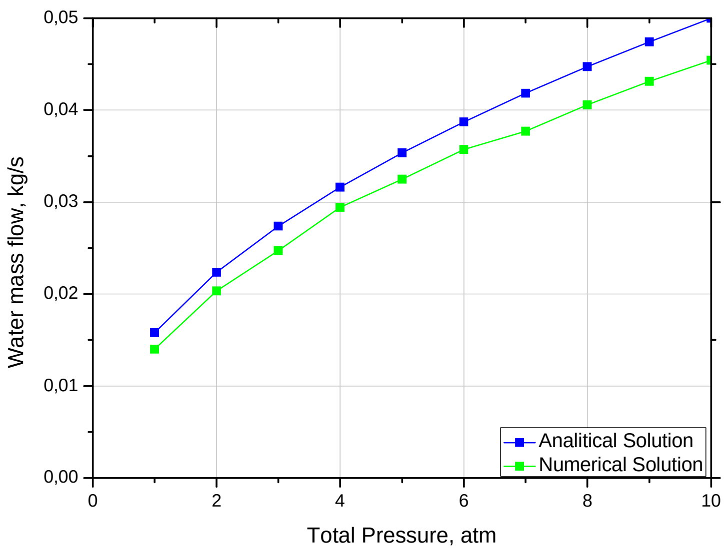
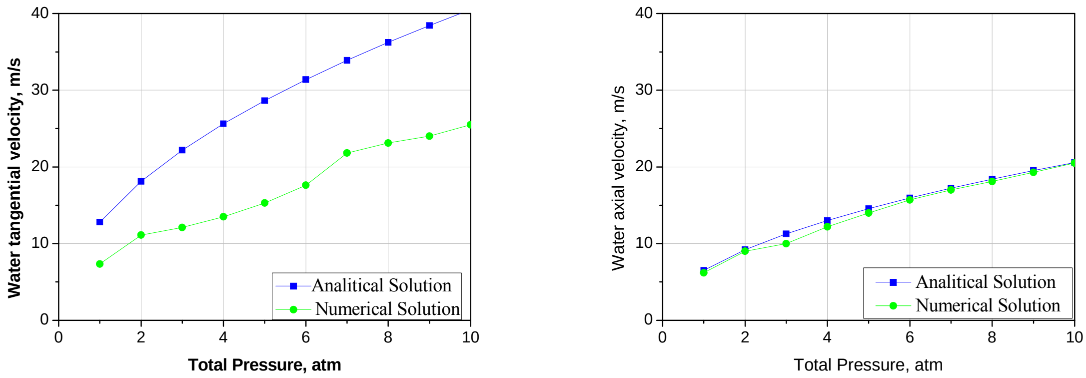


Figure 12. Comparison the analytical results for water mass flow with ones obtained by numerical simulation.



a) b)

Figure 13. Comparison the analytical results for water tangential velocity *a)* and axial velocity *b)* with ones obtained by numerical simulation.

2.4 Conclusion.

The numerical simulation of the tow-phase flow was performed by *k-ε* turbulence model. The boundary layer on the walls and on interface between two fluids was possible to characterize due the good refinement of the mesh in these regions. The analytical solution for axial, tangential velocities and mass flow on the exit of nozzle were obtained. The numerical and analytical results for axial velocity and mass flow present a good agreement as shown in Fig. 12 and. 13.*b)*. The analytical and numerical results for water tangential velocity presented on Fig. 13.*a)* have significant deviation, because in the analytical calculation the friction losses are neglected. The angular momentum in the nozzle is smaller than in the swirl chamber. It affects the calculation of the pulverization angle. The angle of the liquid cone on the exit of the atomizer will have a smaller value than value calculated analytically. The numerical results for the distribution of the mass flow on the transversal section localized on the exit of nozzle shown on Fig. 11 confirm the experimental results presented by Vasiliev *et al*. (1993), Khavkin (2004) and others.

3. ACKNOWLEDGEMENTS

The authors wish to thank the financial support of the National Research Committee – CNPq.

4. REFERENCES

Abramovich, G. N., 1944,"The Theory of the Swirl Atomizer.", In industrial Aerodynamics, Moscow: BNT ZAGI.

Doble, S. M., and Halton E.M., 1947,"The Application of Cyclone Theory to Centrifugal Spray Nozzles", Institute of Mechanical Ingineers, vol. 154.

K.G. Hansen and J. Madsen, 2001, "A Computational and Experimental Study of the Internal Flow in a Scaled Pressure-Swirl Atomizer", M. Sc. thesis, Aalborg University Esbjerg, Denmark.

Khavkin Y. I., 2004, "Theory and Practice of Swirl Atomizers", by Taylor & Francis, New York.

Kliachko, L.A. 1962, "To the Theory of the Swirl Atomizer", teploenergetica, No. 3.

Shanwu W., 2007,Yang V., Hsiao G., Hsieh S. and Mongia C., "Large-eddy simulations of gas-turbine swirl injector flow dynamics", J. Fluid Mech., vol. 583, pp. 99-122.

Vasiliev A. P., 1993, Kudriavtcev V. M., Kuznetcsov V. A., "Theory and calculos of liquid propelant rocket motors", High School.

5. RESPONSIBILITY NOTICE

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