23rd ABCM International Congress of Mechanical Engineering
December 6-11, 2015, Rio de Janeiro, RJ, Brazil

# VALIDATION OF TURBULENCE MODELS IN TURBULENT SPRAYS FORMED FROM A BLURRY INJECTOR

**Bruna Oliveira Passos e Silva Siqueira**
Institute of Mechanical Engineering, Federal University of Itajubá, Av. Bps, 1303, CEP. 37500-903, Itajubá, MG, Brazil.
bruna.passos@gmail.com

**Marco Antônio Rosa do Nascimento**
Institute of Mechanical Engineering, Federal University of Itajubá, Av. Bps, 1303, CEP. 37500-903, Itajubá, MG, Brazil.
marcoantonio@unifei.edu.br

**Cláudia Gonçalves de Azevedo**
National Institute for Space Research, Associate Laboratory of Combustion and Propulsion, Rod. Presidente Dutra, km 40, Cachoeira Paulista, SP, CEP 12630-000
claudia@lcp.inpe.br

**Fernando de Souza Costa**
National Institute for Space Research, Associate Laboratory of Combustion and Propulsion, Rod. Presidente Dutra, km 40, Cachoeira Paulista, SP, CEP 12630-000
fernando@lcp.inpe.br

***Abstract.*** *The concern about environmental issues has led researchers to develop new technologies to reduction in emissions of pollutants and improves the efficiency of combustion processes. The flameless combustion of liquid biofuels sprays has shown a viable technology to achieve these purposes. The numerical modeling has been aiding the development and the understanding of phenomena involved in these processes enabling the advancement of this new technology. This paper presents the numerical modeling of turbulent sprays formed from a blurry injector, which will be used in the simulation of flameless combustion of ethanol. The simulations were performed in ANSYS CFX and the test fluids used in the spray formation was ethanol and air.*

***Keywords:*** *Numerical modeling, turbulent sprays, blurry injector, ANSYS CFX, droplets*

## 1. INTRODUCTION

It is very common to use liquid fuels in processes where the combustion is used as energy source. In most of these systems the fuel is injected in the form of turbulent sprays whose characteristics directly influence the combustion efficiency.

The spray is obtained when the liquid fuel is atomized by passing through a particular type of injector. Atomization is a physical process where the liquid fuel is transformed into a group of small fluid particles. An effective atomization produces a suitable distribution and penetration of droplets into the atmosphere from combustion ensuring greater efficiency in these processes and also contributes to reduction in the emission of pollutants such as CO and NOx.

The search for more efficient atomization of liquid fuels has led several researchers to study both experimentally and numerically the fuel injectors in order to understand how these geometry influences the atomization process. The main objectives of these studies is to obtain information about changes that occur in the liquis and air properties when they pass through the atomizer and the influence of the geometry of the atomizer in the distribution of droplets produced in spray (Hoeg *et al*., 2008; Bolszo and Mc-Donell, 2009; Grech *et al*., 2012).

A new atomization technology called flow-blurry atomizer, recently developed by Gañán-Calvo (2005) has attracted the attention of researchers from the combustion area. The author reports that this atomization technology is simple, robust and reproducible and is capable of reproducing a gas-fluid interaction with high efficiency.

Simmons and Agrawal (2011a) compared the performance of the air-blast injector and flow blurry injector and concluded that the blurry injector obtained a superior performance to the air-blast, playing a most refined spray with droplets of smaller diameters. The authors also noted that the blurry injector reaches peaks of higher axial velocity than those produced by air-blast, which promotes high turbulent interaction generating greater dispersion of gas in liquid phase and a lower pressure droplet in the air line of atomization.

Based on these studies Azevedo (2013) developed a compact flameless combustion system for burning biofuels using the blurry injector. The author investigated the effects of geometry and operating conditions on injector performance. Several characteristics of the generated spray were analyzed as mean diameter of drops, speed distribution in the spray injection region, pressure distribution and other parameters for characterization and understanding of the phenomena present in the atomization of the liquid fuel and the interaction between the continuous and dispersed phase.

The aim of this study is to collect the values of the flow velocity field, the spray cone angle and analyze the distribution of droplets to validate the numerical results with experimental data and to investigate the influence of turbulence models available in ANSYS CFX in the formation process of turbulent spray.

## 2. OPERATING MECHANISM OF FLOW BLURRY INJECTOR

The blurry injector consists of an inner tube for liquid injection surrounded by an outer casing where gas is injected. In Figure 1 is represented the injector geometry, where it is observed a dimension, H, corresponding to the distance between the outlet of the liquid feed tube and the outlet orifice having the same diameter, D. The ratio between these two parameters leads to a new geometric parameter $\psi$ defined as $\psi = H / D$ originating a cylindrical side passage through which flows the gas will assist in the formation of spray. This parameter defines the operating mechanism of the injector. The cylindrical side passage area is equal to the area of outlet orifice when $\psi = H / D = 0.25$.

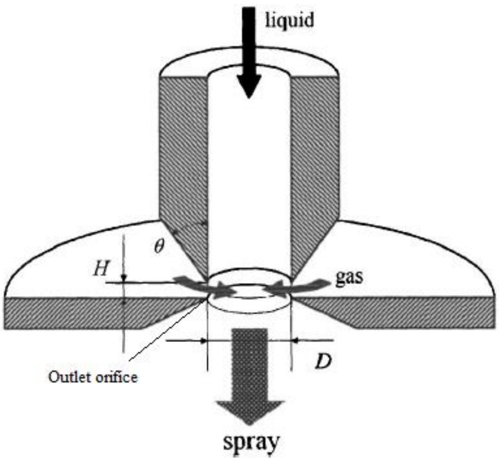



Figure 1 – Simplified schematic of the geometry of the flow blurry injector (Gañán-Calvo, 2005).

In summarized form the operating mechanism of the blurry injector occurs as follows: the liquid is forced through the tube while the gas is forced to pass by cylindrical side passage, a spray results between the two phases and exits the injector through the outlet orifice. The geometric parameter $\psi$ directly influences the characteristics of the formed spray.

The flow blurry effect is observed only for values of $\psi$ smaller than 0.25. For these values, the air flow becomes radial, perpendicular to the axis of symmetry and a stagnation point is formed between the tube exit and the outlet orifice, so part of the gas passing through the cylindrical side passage is forced to flow into within the liquid tube forming a turbulent mixing which results in bubbles generating a biphasic mixture into the liquid injector. This biphasic mixture undergoes a pressure loss while passing through the outlet orifice where occurs the expansion of the gas that explodes the bubbles and disintegrate the liquid into a fine spray.

For values of $\psi$ greater than 0.25 the flow blurry effect is not observed, the flow develops in the flow focusing mode occurring the formation of a micro-jet of fluid that may break symmetrically or asymmetrically resulting in polydisperse spray.

Azevedo (2013) developed in his doctoral thesis three injectors of blurry type by changing the geometry of the nozzle in the region of the injector outlet orifice in order to evaluate the influence of divergent nozzle in the characteristics of the formed spray. It is noteworthy that geometric parameter $\psi$ was kept at 0.25 for the three settings. However, for fuel injection into the combustion chamber, also developed by the author, the conical prototype $n_3$ was chosen, schematized in Figure 2, due to the better distribution of the spray generated by this.

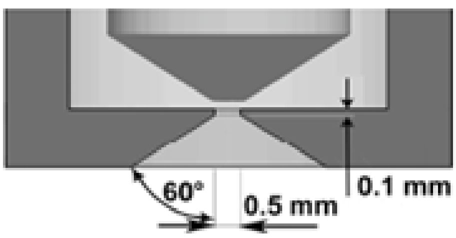



Figure 2 – Conical prototype $n_3$ developed by Azevedo (2013).

Initially Azevedo (2013) defined the operating range of the injector to be used in the injection system considering the variation of the air-liquid mass ratio (ALR = $\dot{m}_{ar}/\dot{m}_{l}$). The ALR parameter was varied in order to investigate its influence on the formation of the spray. The fuels used in the experiments were water, hydrated ethanol and soybean biodiesel. To study in this paper will be considered only results available for hydrated ethanol.

For the prototype n3 the ALR ranges where the effect blurry is observed are shown in Table 1. Under these operating ranges there is a fully formed spray, with good quality and generating very small droplets. For ALR values below the minimum limit a jet of liquid is observed in the spray center line near the injector outlet, as shown in Figure 3. For values above the maximum maximum limit occurs injector clogging due to the large flow of air atomization.

Figure 3 - Jet of liquid in the spray center line for ALR values below the minimum limit presented in Table 1. Azevedo (2013)

Table 1 - Operating range for the nozzle n3 according to the parameter ALR (Azevedo, 2013).

| Liquid | Liquid Mass Flow (g/s) | Air Mass Flow (g/s) | ALR |
|---|---|---|---|
| Water | 0.12 - 0.47 | 0.081 - 0.238 | 0.20 - 1.83 |
| Ethanol | 0.08 - 0.42 | 0.082 - 0.240 | 0.21 - 2.82 |
| Biodiesel | 0.11 - 0.56 | 0.029 - 0.107 | 0.05 - 0.93 |

Determined the injector operating range Azevedo (2013) surveyed the values of air and liquid injection pressure, the air velocity, the average diameter of the droplets and the spray cone angle, important parameters to characterize the injector which influence on the processes of mixing and combustion.

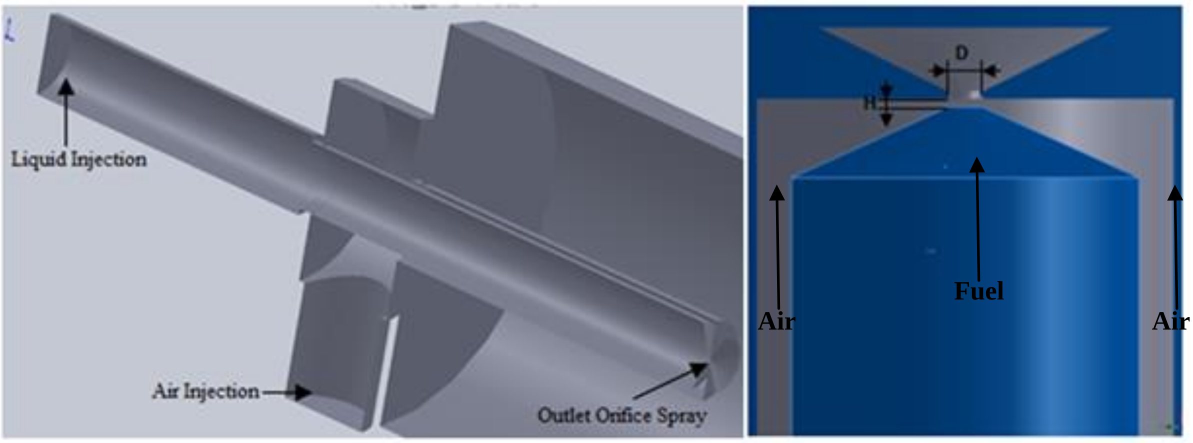
Based on the experimental data will be presented in the following sections of this paper the numerical results in order to evaluate the potential of turbulence models available in ANSYS CFX tool in the reproduction of the flow field of a blurry injector. The numerical results of these simulations will be used in future numerical analyzes using this type injector in a combustion chamber operating in flameless combustion regime.

## 3. VALIDATION OF NUMERICAL RESULTS OBTAINED BY ANSYS CFX FOR GUN FLOW BLURRY

This section presents results from numerical simulations developed by ANSYS CFX 13.0 tool for validation with experimental data available in Azevedo (2013). The aim of this study is to analyze the flow behavior formed downstream of the flow blurry injector and confront the numerical characteristics that result from turbulence modeling through the k-ε standard models, RNG k-ε and SSG Reynolds Stress model and thus identify which models is more efficient in the reproduction of the spray generated by the injector in question.

The computational time spent for each numerical simulation in an common computer with 16 GB of RAM and Intel Core i5-650 3.20 GHz is approximately 50 hours for 1000 iterations.

Were simulated data on injection of hydrated ethanol to ALR = 0.75 ($\dot{m}_{ar}$ = 0.12 g/s e $\dot{m}_{l}$ = 0.16 g/s) and SMD = 8.64 µm (Sauter Mean Diameter). The injector geometry is shown in Figure 4.a generated in SolidWorks 2010 software corresponding to the nozzle n3 developed by Azevedo (2013). Figure 4.b shows in detail the nozzle exit area where H = 0.125 mm and D = 0.5 mm. The tetrahedral mesh contains a total number of 10,517,958 elements and 1,767,247 nodes and was generated in ICEM CFD 13.0. In Figure 5 is outlined the mesh in injector region. The question of independence mesh is not taken into account in this work, because the injector structure requires a very meticulous refinement due to the size of the geometry.




(a) (b)

Figure 4 – Geometry of the flow blurry injector: (a) complete vision of the injector; (b) Detail of the injection region.

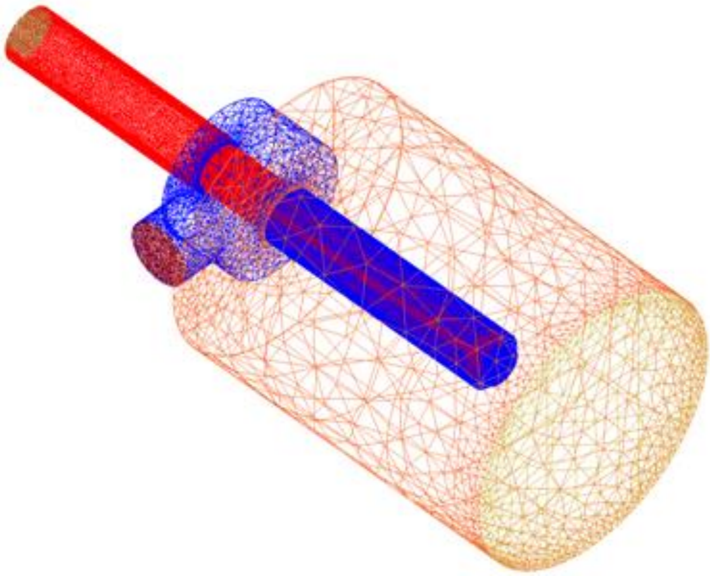


Figure 5 – Detail of the mesh in the injection region.

Three turbulence models available in ANSYS CFX were chosen to perform the numerical simulations: the k-ε standard, the RNG k-ε and SSG RSM. The modeling of turbulence through the k-ε standard is an initial approach to understand the behavior of the injector and also for being a model that produces satisfactory results and achieve more quickly the convergence. The RNG k-ε model and SSG RSM were used for its applicability to flows involving turbulent sprays and they are more accurate than the k-ε standard model.

In Table 2 are the experimental values for ALR = 0.75 and the numerical results related to this work. The parameters used for comparative purposes are the air velocity ($V_{air}$) measured in the region adjacent the wall of the outlet orifice, the cone angle and the pressure of the liquid at the outlet orifice ($P_{liq}$).

Table 2 - Comparison between experimental and numerical values for ALR = 0.75.

| | Cone Angle (°) | $V_{air}$ (m/s) | $P_{liq}$ (bar) | $P_{liq}$ Deviation |
|---|---|---|---|---|
| Experimental (Azevedo, 2013) | 21 | 213.3 | 1.26 | - |
| k-ε Model | 18 | 190.4 | 1.30 | 3.07 % |
| RNG k-ε Model | 19 | 214.1 | 1.28 | 1.56% |
| SSG RSM Model | 21 | 212.6 | 1.28 | 1,56% |

The results obtained by k-ε turbulence modeling are less accurate compared to the values resulting from other turbulence models. It is noted that the air velocity near the wall of the injector outlet orifice ($V_{air}$ = 190.4 m/s) is much lower than the experimentally measured ($V_{air}$ = 213.3 m/s) and the liquid injection pressure ($P_{liq}$ = 1.30 bar) is larger than the experimental ($P_{liq}$ = 1.26 bar). An analysis of Figure 6 shows that the k-ε model generates a spray of low penetration and promotes low recirculation of the fuel within the combustion chamber which can be a problem when the combustion reactions are included in the simulations negatively influencing the performance of process. Hovenden and Davidson (1997) showed in their work about combustion of turbulent sprays that the model k-ε standard is not adequate in predicting velocity profiles in the injectors exit because this model is very diffusive and results in much flatter velocity profiles than those really obtained from the spray.

For simulations with turbulence modeling by RNG k-ε model results are much closer to the real ones. The value of the air velocity encountered, $V_{air}$ = 214.1 (m/s), is only 0.3% higher than the experimental velocity, $V_{air}$ = 213.3 (m/s), the value of liquid injection pressure, $P_{liq}$ = 1.28 (bar), is 1.5% higher than the experimental, $P_{liq}$ = 1.26 (bar). These values can be considered fairly accurate, especially when analyzing the velocity field inside the combustion chamber in Figure 7, where we can observe the spray penetration increased due to the higher air velocity that accelerates the fuel particles increasing its dispersion in the chamber and promotes greater recirculation rate that is essential to achieve the flameless combustion regime.

The SSG RSM model was chosen for analyzes to be an extension of the Reynolds Stress Model, and by some authors as Rochaya (2007), Li et al. (2014) and Jamali (2014) emphasize the best prediction of the properties of turbulent sprays. The results obtained in the simulations of this study confirm this fact, the cone angle reproduced by the model was the same as obtained in the experiment of Azevedo (2013). The air velocity at the outlet orifice equals $V_{air}$ = 212.6 (m/s), only 0.5% lower than the experimental value. The value of liquid injection pressure was equal to that obtained in the RNG k-ε model, $P_{liq}$ = 1.28 (bar). In Figure 8 note that the SSG RSM model reproduces a spray with well-defined cone angle and high penetration in the chamber regions are still observed with high vorticity distributed throughout the domain which promotes better distribution of the fuel due to high recirculation of the air/fuel mixture, this feature very important when working with the flameless combustion. In general this model proves to be much more feasible than the models k-ε standard and RNG k-ε in the representation of spray generated by the injector flow blurry.

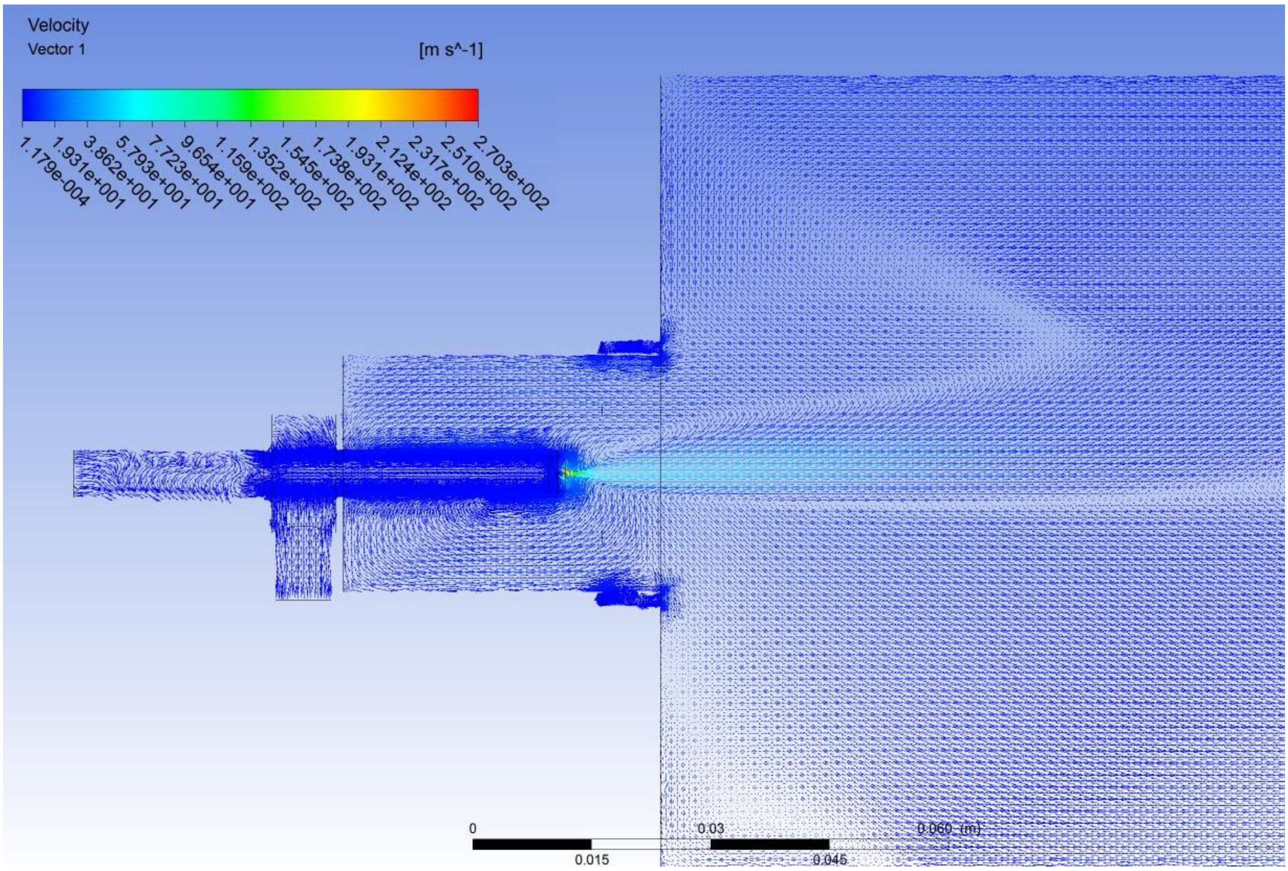



Figure 6 - Velocity field distribution within the combustion chamber generated by the k-ε model.

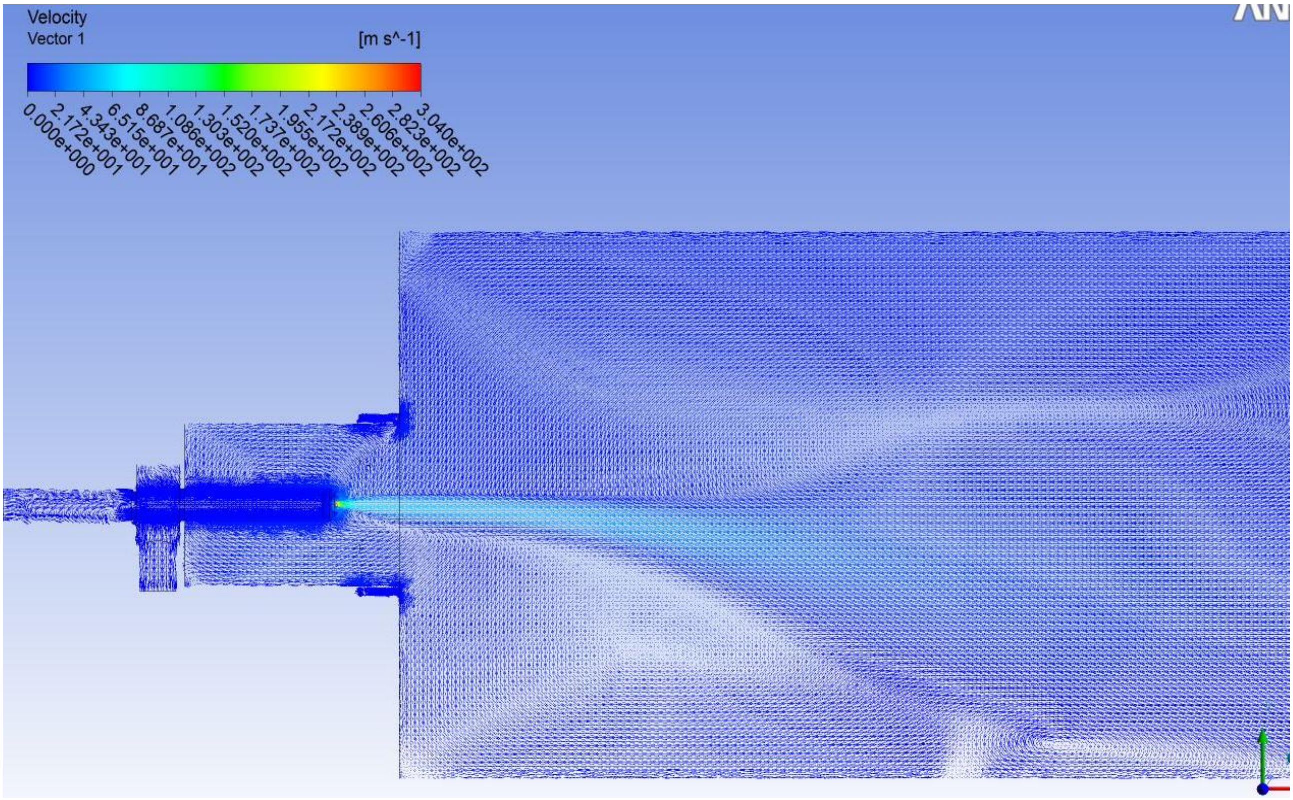



Figure 7 - Velocity field distribution within the combustion chamber generated by the RNG k-ε model.

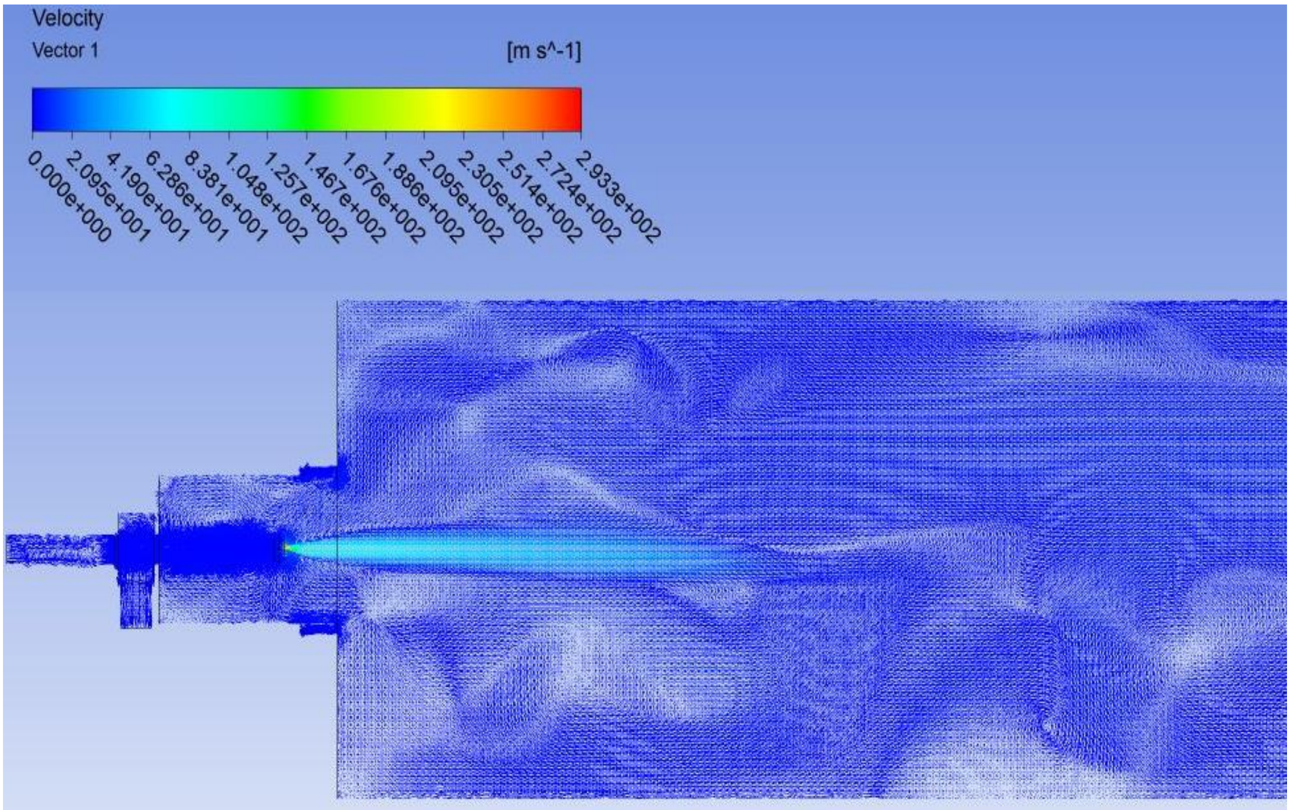
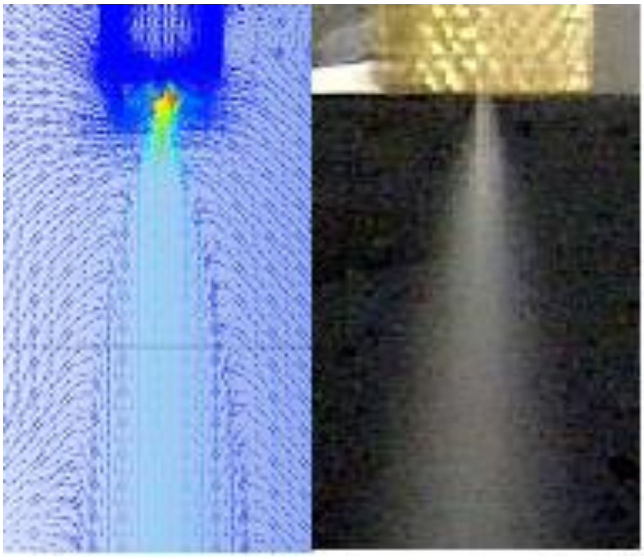



Figure 8 - Velocity field distribution within the combustion chamber generated by the SSG RSM model.

In Figure 9 are compared sprays formed in the numerical simulation by SSG RSM (Fig. 8.a) and in the experiment of Azevedo (2013) (Fig. 8.b) showing the potential of this numerical model to simulate the flow injector blurry.



(a) (b)

Figure 9 - Comparison between sprays formed in the flow blurry injector: (a) modeling through SSG RSM; (b) Experimental (Azevedo, 2013).

The last parameter for analysis in this study is the distribution of drops inside the combustion chamber. The investigations showed that the droplets are concentrated in the central region of the spray and the phenomenon of droplets break-up was observed from the reduction of droplet diameter as provided in Figure 10. However, to ensure greater precision and certainty in the results a deeper analysis is needed in order to analyze the droplets evaporation modeling and also increase the number of parcels in the domain representing the injection of drops, which will be held in future work.

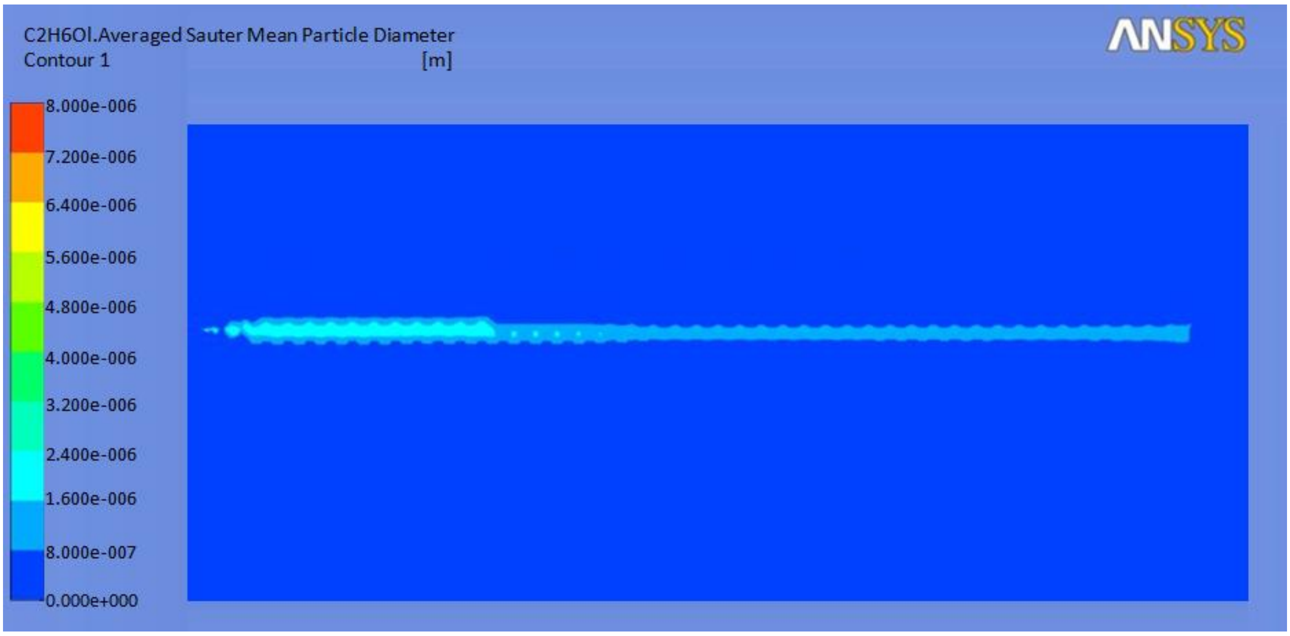



Figure 10 - Distribution of ethanol droplets within the combustion chamber generated by the SSG RSM.

## 4. CONCLUSIONS

In the present study the spray formed by a flow blurry injector was numerically simulated and validated with experimental data from Azevedo (2013). The aim of the work was to evaluate the potential of turbulence models available in ANSYS CFX in the reproduction of the spray characteristics. The k-ε model does not reproduced satisfactorily the velocity distribution within the chamber, producing a spray of low quality and unable to generate recirculation regions of air/fuel mixture, which will negatively influence the flameless combustion process.

The RNG k-ε model and SSG RSM reproduced with high precision the experimental results, because these models have a most appropriate formulation for the calculation of the eddy viscosity and also take into account the effect of swirl on turbulence, a very common phenomenon in turbulent sprays. The high recirculation generated by these models is fundamental to reach the flameless combustion regime, mainly for the SSG RSM model that reproduced a spray with good quality and high vorticity distribution within the chamber.

Regarding the distribution of the droplets within the combustion chamber is necessary to develop new simulations varying parameters such as particle number entered in the domain, simulation time and a deeper analysis of the modeling of droplet evaporation.

The information obtained in this work will be used as a basis in developing new numerical simulations to investigate the flameless combustion regime.

## 5. ACKNOWLEDGEMENTS

The authors are grateful to CAPES, FAPEMIG and CNPq for financial help and ESSS for technical support which contributed to the development of this work.

## 6. REFERENCES

Azevedo, C. G., 2013. *Development of a compact system for flameless combustion using a blurry injector for burning biofuels*. PhD thesis, INPE, São José dos Campos.

Bolzo, C. D., Mc-Donnel, V. G., 2009. "Evaluation of plain-jet air blast atomization and evaporation of alternative fuels in a small gas turbine engine application". *Atomization and Sprays*, Vol. 19, n. 8, p.771–785.

Gañán-Calvo, A. M., 2005. "Enhanced liquid atomization: from flow-focusing to flowblurring". *Applied Physics Letters*, Vol. 86.

Grech, N., Mehdi,A., Zachos, P. K., Pachidis, V., Singh, R., 2012. "Effect of combustor geometry on performance of airblast atomizer under sub-atmospheric conditions". *Enginneering Aplications of Computational Fluid Mechanics*, Vol. 6, p.203-213.

Hoeg, D. P., Wang, Z., Friedman, P. D., Laoulache, R. N., 2008. "Investigation of a coaxial air-blast atomizer using particle image velocimetry and computational fluid dynamics". *Atomization and Sprays*, Vol. 18, n. 8, p. 739–759.

Hovenden, R. J., Davidson, M. R., 1997. "Turbulence modelling of single phase flow in a spray dryer". In *Proc.of Inter. Conf. on CFD in Mineral and Metal Processing and Power Generation,* University of Melbourne, Parkville, Victoria.

Jamali, S. H., 2014. *Computational modeling of turbulent ethanol spray flames in a hot diluted coflow.* Master of Science in Mechanical Engineering Thesis, Delft University of Technology.

Li, K., Zhou, L., Chan, C. K., 2014. "Large-eddy simulation of ethanol spray-air combustion and its experimental validation". *Chinese Journal of Chemical Engineering*, Vol. 22(2), p.214-220.

Rochaya, D., 2007. *Numerical simulation of spray combustion using bio-mass derived liquid fuels.* PhD thesis, Cranfield.

Simmons, B.M., Agrawal, A.K., 2011a. "Spray characteristics of a flow-blurring atomizer". *Atomization and Sprays*, Vol. 20, n. 9, p.821-825.

## 7. RESPONSIBILITY NOTICE

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