Computational Research Progress in Applied Science & Engineering

ISSN 2423-4591

CRPASE Vol. 04(01), 1-5, March 2018

Numerical Analysis of the 3D Flow in Swirl Injector with Spiral Paths

Kiumars Khani Aminjan [a*], Mohammad Mahdi Heydari [a], Esmail Valizadeh [b]

[a] *Aerospace University Complex, Malek Ashtar University of Technology, Tehran, Iran*
[b] *Faculty of Aerospace Engineering, K. N. Toosi University* of Technology*, Tehran, Iran*

Keywords

Swirl injector,
Spiral path,
Spray angle,
Droplets diameter.

Abstract

Due to the significant impact of the injectors on the efficiency and durability of the engine, the design and simulation of the injector is very important. In this paper at first the design and computation of a swirl injector with spiral paths is proposed. The injector is designed in such a way that it has an open spraying angle and very low layer thickness which is suitable for limiting the length of the chamber and provides a finer powder. Since the phenomenon of creating and developing an air cavity in the internal flow of injectors and simulating it which is due to the presence of two turbulent twist currents in two different phases with a common free surface is complicated, the internal flow has a potential effect on the spray properties of the swirl injectors. Therefore, in the next step, simulation of the internal flow of the injector with two different fluids (water and kerosene) has been investigated and parameters such as spray angle, discharge coefficient, speed coefficient, spray film thickness, and average diameter of the droplets have been investigated. The Fluent® software is used for the present simulation and for a two-phase flow, the volume of fluid (VOF) method is considered. Further to these, the flow turbulence is simulated with the RNG model. since the results of the simulation have a very small error with design hypotheses, the results of simulation can be used in various industrial applications, especially in the turbine engines industry with the aim of reducing costs.

d': The average diameter of the droplets in the first phase of spraying
d'': The average diameter of the droplets in the second phase of spraying
K_v: Speed coefficient
α: Spray angle
μ : Discharge coefficient
μ_L: Dynamic viscosity
ν : Kinematic viscosity
t: The thickness of the film
D_s: The diameter of the swirl chamber
L_s: The length of the swirl chamber
L_o: The length of the orefis
D_o: The diameter of the output hole
A_{in}: Spiral paths area
K: Injector constant

1. Introduction

Due to the high impact on performance, efficiency and stability of the engine, the design or choice of injector types is one of the most important steps in engine design. The size of the injector should be chosen in such a way as to provide the required mass flow rate and differential pressure. On the injectors, the fluid is fed into the injector from one side and through the supply system, and on the other side it is discharged through one hole. This exit must be such that the propellant is atomized to optimal combustion and converted to steam. In other words, the injector is a device for converting a fluid into droplets and distributing these particles in the combustion chamber space. To the extent that the particle breakdown is performed faster and better, the evaporation of the fluid particles will be carried out at a higher rate, resulting in higher thermal efficiency and increased the thrust force. Therefore, the injection of fuel into the combustion chamber is a very important issue in the combustion process.

---

[*] Corresponding Author:
E-mail address: kiomars.khani67@gmail.com – Tel, (+98) 9117222988

Received: 12 November 2017; Accepted: 28 February 2018

Due to the complexity of the spray process, the research has often been carried out experimentally. One of the most important steps in understanding the process of powdering or atomization of fluid and providing different failure models is to investigate the spraying process using analytical and empirical methods. The process of forming and preparing a mixture of fuels has a significant effect on the combustion process and has a direct effect on the thermal efficiency and the amount of propulsion. The spray pattern and spray characteristics of the injector influence the formation of the fuel mixture and affect the combustion properties in a wide range of engine operating conditions.

Due to the advantages of swirl injectors compared with other types of injectors and the increased use of these types of injectors, the main goal of this study is to investigate the spray behavior of a swirl injector with spiral path such as spraying con angle, fluid thickness, discharge coefficient and etc. In this type of injectors, the flow is axially introduced into the injector, which, by flow through the spiral path, causes the flow to occur due to the radial velocity, causing the formation of a hollow conical spray. The information on this type of injector is very limited, and this has led to a more thorough research into the design and simulation of spiral type injectors.

2. Literature Background

Considering the importance of swirl injectors, this issue has always been of interest to researchers. Although the design process and flow physics in the swirl injectors with a spiral path and centrifugal injectors are very much the same, but as discussed in the previous section, in relation to swirl injectors with a spiral path have limited information and most of the studies are about tangential inlet tangential injectors.

Lefebvre [1] has organized the most important references on atomization and sprays, including some aspects of pressure-swirl atomizers design procedures and presented some predictions on discharge coefficients, spray-cone angles and mean droplet sizes. The work of Couto et al. [2] showed a theoretical formulation for estimating the Sauter Mean Diameter (SMD) of droplets generated by pressure swirl atomizers. This was done by extending the model of Dombrowski and Johns [3] on the disintegration of viscous liquid sheets generated by fan-spray atomizers, the results comparing satisfactorily with available experimental data and other existing empirical models. Bazarov and Yang [4] have discussed the liquidpropellant rocket engine pressure-swirl atomizer dynamics and its relation with flow oscillations. Paula Souza [5] presented a design procedure and performed an experimental analysis of a coaxial pressure-swirl bi-propellant atomizer for liquid-propellant rocket engines. Jones [6] presented a design optimization of a large pressure-jet atomizer for furnace power plants. Lefebvre [7] discussed the application of pressure-swirl atomizers in gas turbine combustion chambers.

Morad et al. [8] studied the spray characteristics of a liquid-liquid coaxial swirl atomizer at different mass flow rates. They measured the Sauter mean diameter, and droplet axial and radial velocities using a system of PDA. Their study showed that smaller particles possess a smaller magnitude of velocity in contrast to larger particles which have a greater ones. Their study also revealed a peak in the velocity magnitude of particles in the radial coordinates of the spray. In another study by Radke [9] on a liquid-liquid swirl injector, further insight into geometric and flow parameters has been provided. The output diameter of an atomizer and flow properties have significant effects on sheet and ligament breakup. He supposed that increasing the Reynolds effect on SMD is similar to increasing the output diameter. Ibrahim et al. [10], further delved into instability mechanisms and the breakup of liquid sheets. Increasing the viscosity reduces the radial and tangential velocity components and the cone angle of a spray as a consequence. There are different experimental methods to define the Sauter mean diameter and study the spray characteristics. Optical drop sizing methods and image processing techniques are very popular and often desired. The complexity of the atomization process makes the prediction difficult and the results are still reported in terms of various correlations. Shafaee et al. [11] used an image processing technique to develop a correlation between the spray cone angle and the Weber number. By using other methods, Merington and Richardson [12] proposed a correlation for the Sauter mean diameter of spray produced by a circular orifice atomizer.

3. Injector Design

The design objective is to determine the dimensions of a swirl injector based on the available initial information including Q (volumetric flow rate) or G (mass flow rate), α (spray angle), ΔP (pressure difference), ρ (density), and ν (kinematic viscosity). First, the initial dimensions are calculated according to the geometric constant K, and then the other dimensions are determined accordingly. This is not unique, since identical K can be different from the choice of R, i, d_p and d_o. In general, the design algorithm for swirl injectors is as follows:

Information such as Q or G, α, ΔP, ρ, and ν should be given.

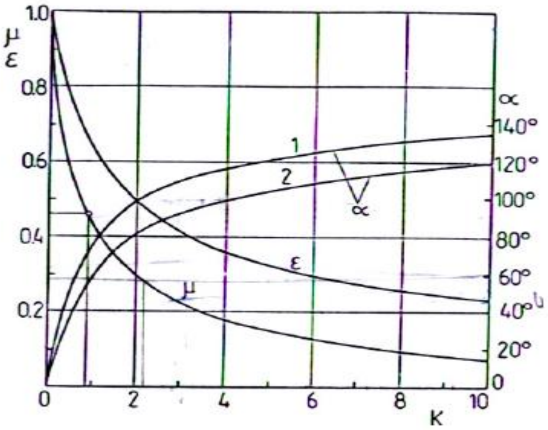

Figure 1. The relationship between the spray angle and discharge coefficient and the filling efficiency of the hole with injector constant [13]

a) Using the curve of Figure 1, the values of K and μ are found.

b) Using Eq. (1), the diameter of the output hole is obtained as

$$d_o = \sqrt{\frac{4G}{\pi\mu\sqrt{2\rho P_t}}} \qquad (1)$$

c) The inlet hole diameter is found by assumptions for *i* and *R*, usually $R=(2\text{-}5)r_o$ or by using the Eq. (2). In the cases where the shape of the holes is not a circle, instead of d_p, the value of A_p is determined.

$$d_p = \sqrt{\frac{2Rd_0}{iK}} \qquad (2)$$

d) If the conditions below are met, then one can ignore the effects of viscoseity.

$$B = \frac{R}{r_p} \qquad (3)$$

$$\frac{B^2}{i} - K \leq \frac{2}{\lambda}(\Phi^{1.5} - 1) \qquad (4)$$

e) If the effects of viscoseity is not ignored, then, one should use k_λ (Eq. (5)) instead of *K* in the equations and retrieve the values of α and μ from the graph again.

$$k_\lambda = \frac{Rr_0}{ir_p^2 + \left(\lambda/2\right)R(R - r_0\ )} \qquad (5)$$

f) For other sizes, the diameter of the swirl chamber, the length of the orefis and the inlet diameter of the fluid inlet would be obtaines as

$$Ds = 2R + d_p \qquad (6)$$

$$lp = (1.5 - 3)d'_p \qquad (7)$$

$$\varphi = \frac{A_p}{A'_p} = (\frac{d_p}{d'_p})^2 \qquad (8)$$

where, $\varphi = 0.85\text{-}0.9$ and $d'_p = \frac{d_p}{\sqrt{\varphi}}$.

The calculation method is not limited to injectors with a tangential inlet hole and includes injectors with a spiral inlet. After step by step the above steps, the injector are designed and changing the parameters for optimization is within the permissible range of the designer.

The information and some key uncertainties in the design of the injectors studied in this paper, which are calculated using the above method, are presented in Tables 1 and 2, respectively.

Table 1. Design parameters

Quantity	Amount	Unit
α	100	deg
μ	0.17	-
ν	1.005×10^{-6}	m^2/s
G	10	gr/s
P	998.2	kg/m^3
Φ	0.88	-

Table 2. Design results

Parameter	Quantity
d_o	1.84 (mm)
D_s	5 (mm)
l_o	0.8 (mm)
l_s	6 (mm)
A_{in}	0.25 (mm^2)

4. Modeling of Swirl Injector with Spiral Paths

In this type of injector, the flow is axially introduced into the injector and spiral path causes radial velocity and tangential velocity which results in the formation of a hollow conical spray. The injector geometry modeling is done 3D and with gambit software, it presented in Figure 2.

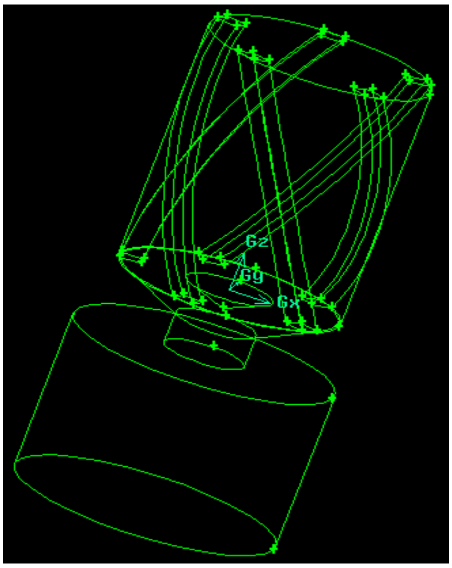

Figure 2. Injector geometry with outlet domain

Grid was created by Ansys software and due to the physical visibility of the fluid flow inside the injector, the injector output and the inverters of the injector must be sufficiently fine-tuned to allow fluid flow to be shown in those areas (Figure 3).

The independence of the results from the numerical resolution to number of cells has been investigated and the number of computational cells is considered to be about 300,000 cells, which indicates the high cost of computing. Independence results from the gride are presented in Table 3 and Figure 4.

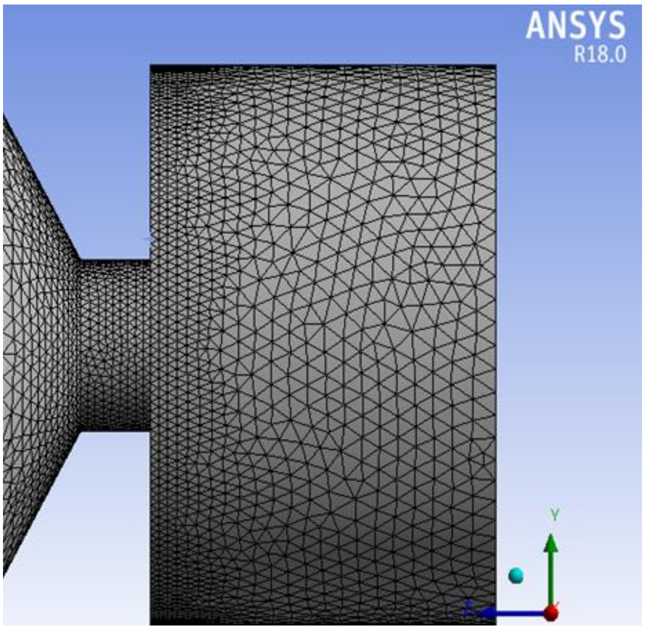

Figure 3. Display meshing at injector outlet

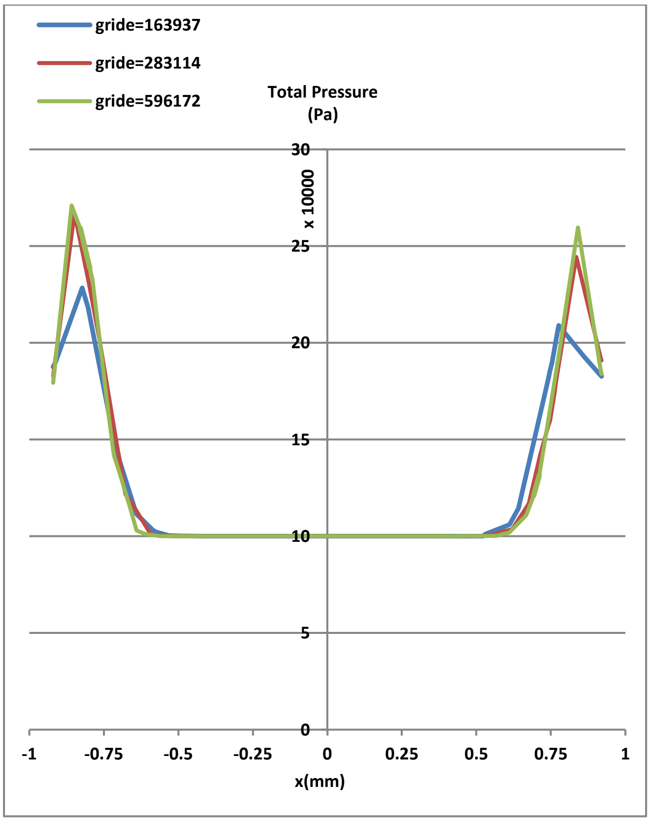

Figure 4. Total pressure for different number of gride

Table 3. Results of gride study

Gride number	μ	t (mm)
163937	0.1641	0.2471
283114	0.1619	0.2455
596172	0.1610	0.2448

5. Numerical Solution

The main governing equations of the flow that are solved by numerical codes are the continuity equations and Navierstock equations solved in an incompressible manner and derived of the energy equation. For internal modeling and analysis, a numerical solution method (Ansys Fluent 18) has been used. For modeling of two-phase flow in the range of two-phase contact area, the VOF method is used. In this technique, it is assumed that the two fluids do not interfere [14].

To solve the momentum equations, a second-order algorithm is used and the equations are explicitly solved. Also, a simple model is used to obtain a discrete equation for correcting pressure in computational cells. The fluid in the simulations is water and kerosene. According to design the inlet flow conditions were 10 gr/s.

As shown by Figure 5, an interesting phenomenon that occurs when the fluid is emitted from the injector is the creation of a Rewind region due to the effect of the induction effect of the flow of fluid upon leaving the orifice, which causes the velocity to flow into the surrounding air, while due to the pressure drop the central portion of the injector, the air inside the injector is sucked. The combination of these effects creates an open air rotation area.

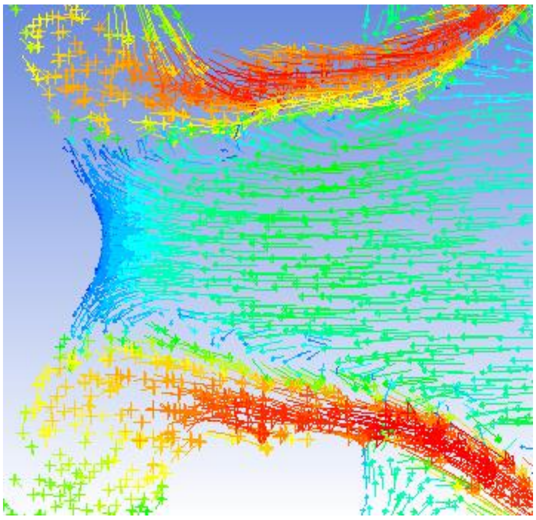

Figure 5. Velocity vector in the injector output

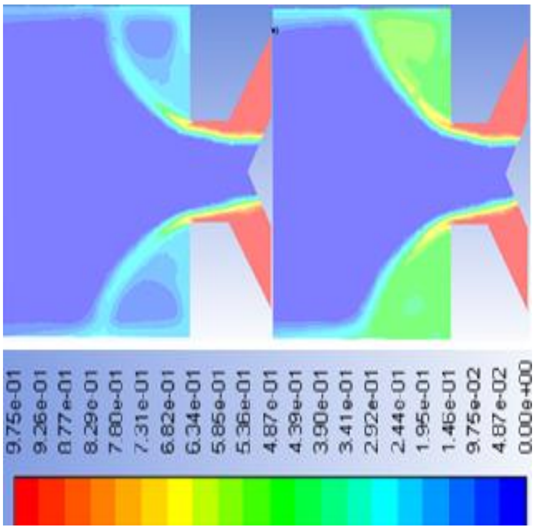

Figure 6. Fluid phase contour at the injector output (water is left and right related to kerosene)

Figure 6 shows that at the outlet of the injector, the central portion has an air core and there is no fluid in the central part, this phenomenon creates a spray in the form of a hollow cone. Comparing Figure 6 and Table 4, the result is that the spray angle for water and kerosene is slightly different.

Table 4. Summary of numerical analysis results

Specifications	Water Liquid	Kerosene Liquid
ν (m^2/s)	1.005×10^{-6}	3.08×10^{-6}
μ_L(kg/ms)	0.001003	0.0024
ρ (Kg/m^3)	998.2	780
α(deg)	90.516	90.798
μ	0.1619	0.1407
t (mm)	0.2455	0.2847
K_v	0.4797	0.5022
$d'(\mu m)$	37.24	61.89
$d''(\mu m)$	6.242	6.466

Increasing density and viscosity reduces speed, but given that the density factor is more effective than viscosity , the total speed for kerosene is more than water (Figure7) and this increases the angle of spraying and the speed coefficient. On the other hand by increasing the viscosity that prevents fluid from flowing, the thickness of the fluid and the discharge coefficient increasing (Figure 8). with increasing viscosity due to increased adhesion, especially near the wall diameter of the droplets increases.

According to the Table 5, the results obtained from the numerical solution have very good match with design assumptions.

Table 5. Comparison of numerical analysis results (CFD) with design assumptions

	Design	CFD	Difference of CFD results with Design
α(deg)	100	95.5	9.5%
μ	0.17	0.15	11.76%

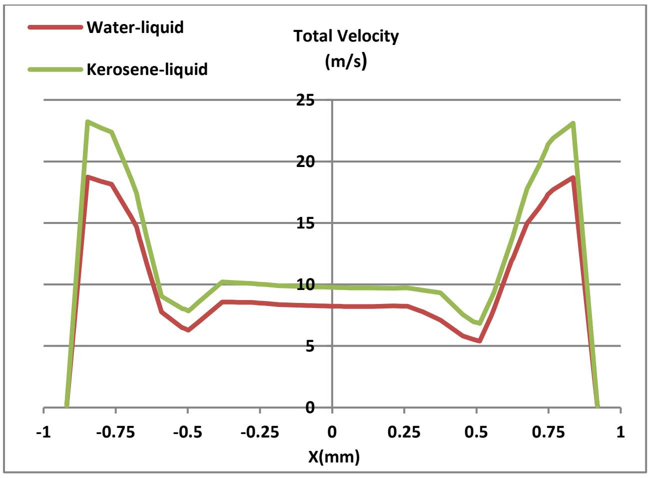

Figure 7. Comparison of the velocity magnitude at the injector output

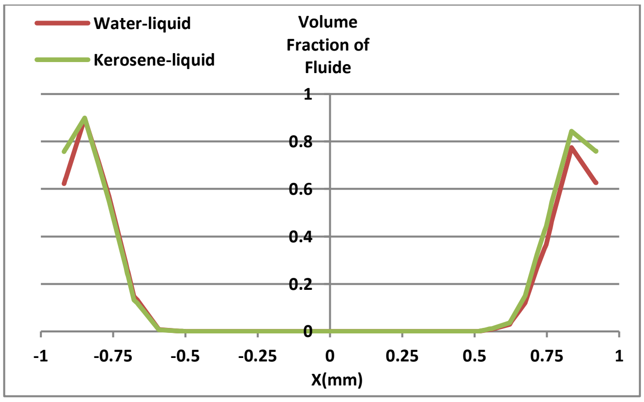

Figure 8. Comparison of the volume fraction of fluid at the injector output

Figures 7 and 8 show that the kerosene droplets are faster than water droplets that suggesting that larger droplets are faster on the other hand, these figurs show that the speed coefficient for kerosene is higher than water and this result is also seen in Table 3.

6. Conclusions

In this paper at first, the swirl injector design algorithm was presented and then using this algorithm and based on a series of initial assumptions, injector design was done. In the next step, the injector designed with water fluid was analyzed numerically and the spose of this analysis was repeated with the kerosene fluid. In the flow survey at the outlet, it was observed that the flow in the output was in the form of a hollow cone. An interesting phenomenon that was observed was the return flow in the central portion of the injector output, this phenomenon results from interactions of velocity and pressure drop in the injector. By repeating the numerical solution for kerosene and comparing its results with the results of water, it is observed that with an increase in the viscosity, the average diameter of the droplets and the thickness of the fluid film increases because with increasing viscosity the surface tensile forces have more resistance to aerodynamic forces and the average diameter of the droplets and the thickness of the fluid film increases as a consequence.

References

[1] A.H. Lefebvre, Atomization and Sprays, Hemisphere Publishing Corporation, New York, 1989.

[2] H.S. Couto, Jr.J.A. Carvalho, D. Bastos-Netto, Theoretical Formulation for Sauter Mean Diameter of Pressure-Swirl Atomizers, Journal of Propulsion and Power 13 (1987) 691–696.

[3] N. Dombrowski, W.R. Johns, The Aerodynamic Instability Disintegration of Viscous Liquid Sheets, Chemical Engineering Science 18 (1963) 203–214.

[4] V.G. Bazarov, V. Yang, Liquid-Propellant Rocket Engine Injector Dynamics, Journal of Propulsion and Power 14 (1998) 797–806.

[5] J.R. Paula Souza, Estudo de um Injetor Centrífugo Bipropelente utilizado em Motor Foguete a Propelente Líquido (Thesis in Portuguese), Instituto Tecnológico de Aeronáutica, São José dos Campos, Brazil, (2001).

[6] A.R. Jones, Design Optimization of a Large Pressure Jet Atomizer for Power Plant, Proceedings of the 2nd International Conference on Liquid Atomization and Spray Systems, Madison, Wis., (1982).

[7] A.H. Lefebvre, Gas Turbine Combustion. Hemisphere Publishing Corporation, New York, 1983.

[8] M.R. Mosad, M.R. Soltani, M. Ashjaee, K. Ghorbanian, Spray characteristics of a liquid-liquid coaxial Swirl a Lomizer at different mass flow rates, Aerospace science and technology 9 (2005) 592–604.

[9] CD. Radke, Experimental characterization of a liquid-liquid coaxial Swirl rocket injector using non-invasive Optical and X-ray techniques, PhD diss, Iowa State University, (2012).

[10] AA. Ibrahim, Comprehensive Study of Internal Flow Field and Linear and Nonlinear instability of an Annular Liquid Sheet Emanating from an AIOmizer, PhD diss, University of Cincinnati, (2006).

[11] M. Shafaee, SA. Banilabaie, W. Esfahanian. M. Ashjaee, An investigation on effect of geometrical parameters on spray Cone angle and drople size distribution of a two-fluid atomizer, Journal of mechanical science and technology 25 (2011) 3047–3052.

[12] A.C. Merington, E.G. Richardson, The break-up of liquid jets, Proceedings of the Physical Society 59 (1947) .

[13] L.P. Bayvel, LiquidAtomization.Taylor & Francis, New York, 1993.

[14] S. Khani, M.A. Moghadam, M. Nikookar, Pressure Fluctuations Investigation on the Curve of Flip Buckets Using Analytical and Numerical Methods, Computational Research Progress in Applied Science & Engineering 3 (2017) 165–171.