CHEMICAL ENGINEERING TRANSACTIONS
VOL. 61, 2017

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Petar S Varbanov, Rongxin Su, Hon Loong Lam, Xia Liu, Jiří J Klemeš

ISBN 978-88-95608-51-8; **ISSN** 2283-9216

DOI: 10.3303/CET1761321

On Flow Regime Inside Plain-Orifice Nozzle

Xinshun Tan, Yang Qingqing, Jiansong Li, Rongshan Bi*, Shiqing Zheng

Resrarch Center for Computer and Chemical Engineering, Qingdao University of Science & Technology, Qingdao 266042, Shandong, China
birongshan@163.com

The plain-orifice is the most common type of atomizer. The flow regime inside plain-orifice nozzle has a tremendous effect on the external spray. At present, the judgement method of the flow pattern is not uniform. The purpose of this study is to numerically investigate the relationship between the change of flow pattern, the ratio of nozzle length and nozzle diameter (L/d), the ratio of nozzle radius of the entrance and nozzle diameter (r/d) in plain-orifice nozzle. The computational fluid dynamics (CFD) code FLUENT was used to perform the numerical simulation of the cavitating flow and hydraulic flip in the nozzles. In this paper, the relationship between the change of flow pattern, r/d and L/d was obtained, and the relation curves between L/d and r/d were plotted, and a new model of flow pattern was obtained. The model established in this paper is more accurate than the empirical model to judge the change of flow pattern. The new model is helpful to modify the existing empirical model. According to the relationship, the researcher can use the method of reasonably adjusting the relationship between the nozzle structure and operating conditions to avoid the generation of hydraulic flip, and to get better atomization effect.

1. Introduction

The plain-orifice is the most common type of atomizer. The plain orifice may operate in three different regimes: single-phase, cavitating and flipped (Hong et al., 2010). When the flow regime inside plain-orifice nozzle is cavitating, the better atomization effect can be obtained. When the flow regime inside plain-orifice nozzle is flipped, the atomization effect is the worst (Payri et al., 2004). Generally, the atomization effect becomes better by increasing the injection pressure, but the flow regime inside plain-orifice nozzle can change from cavitation flow to hydraulic flip with increasing injection pressure when the nozzle structure is not changed (Soteriou et al., 1995). If we want to strengthen atomization effect by increasing the inlet pressure, we need to determine whether the increase in pressure will lead to the flow regime inside plain-orifice nozzle from cavitation to hydraulic flip. So, at higher injection pressure, the researchers can change the nozzle structure so that the flow pattern in the nozzle is cavitating (Ku et al., 2011). However, there is no established theory for determining the flow regime inside plain-orifice nozzle. One must rely on empirical models obtained from experimental data. Generally, when r/d is greater than 0.05, then flip is deemed impossible (FLUENT V6.3 Documentation, 2003). However, Ghassemieh et al. (2006) got that the value of r/d is greater than 0.05, flow regime inside plain-orifice nozzle may be hydraulic flip, which causes the problems in the design of the nozzle. In this paper, a new criterion for judging the occurrence of the hydraulic flip is put forward basing on the analysis of the detailed information of the flow regime inside plain-orifice nozzle by using modern CFD technology.

2. Model setup and solution

2.1 Computational model

The commercial CFD software FLUENT was used to investigate the cavitating flow in the plain-orifice nozzle (Hong et al., 2011). The cavitation model in the FLUENT was based on the so-called full cavitation model developed by Singhal et al. (2002). In order to account for locally transient flow, the realizable k-ε turbulence model was used in this study. Also, the mixture model was used for multiphase cavitating flow and a no-slip condition between the liquid and vapor phases was assumed. The working fluid is assumed to be a mixture of liquid, steam, and noncondensing gas. The mass fraction of steam is determined by the vapor transport equation, continuity equation and momentum conservation equation. This model accommodates also a single-phase formulation where the governing equations is given by:

$$\frac{\partial}{\partial t}\left(f_v \rho\right)+\nabla\cdot\left(f_v \rho \overline{V_v}\right)=\nabla\cdot\left(\Gamma\nabla f_v\right)+R_e+R_c \tag{1}$$

Where, f_v is vapor mass fraction; ρ is the mixture density; $\overline{V_v}$ is the velocity vector of the vapor phase; Γ is diffusion coefficient; R_e and R_c are the vapor generation and condensation rate terms (or phase change rates). The rate expressions are derived from the Rayleigh-Plesset equations, and limiting bubble size considerations (interface surface area per unit volume of vapor). These rates are functions of the instantaneous, local static pressure and are given by:

When $P \le P_v$

$$R_e = F_{vap}\frac{\max(1.0,\sqrt{k})\left(1-f_v-f_g\right)}{\sigma}\rho_l\rho_v\sqrt{\frac{2}{3}\frac{(P_v-P)}{\rho_{ell}}} \tag{2}$$

When $P>P_v$

$$R_c = F_{cond}\frac{\max(1.0,\sqrt{k})f_v}{\sigma}\rho_l\rho_v\sqrt{\frac{2}{3}\frac{(P_v-P)}{\rho_l}} \tag{3}$$

where the suffixes l and v denote the liquid and vapor phases, $\sqrt{k}$ is the local turbulence intensity, σ is the surface tension coefficient of the liquid, P_v is the liquid saturation vapor pressure at the given temperature, and F_{vap} and F_{cond} are empirical constants. The default values are F_{vap} = 0.02 and F_{cond} = 0.01.

2.2 Geometric model

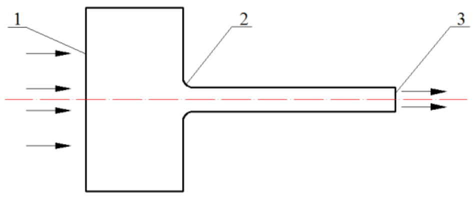

Figure 1: Schematic diagram of nozzle structure. 1 - inlet; 2 - inlet corner; 3 - outlet

The structure of the plain-orifice is relatively simple, as shown in Figure 1. The working fluid is entered into the nozzle through the nozzle inlet with diameter of 15 mm, and the fluid is accelerated at inlet corner. The working fluid is ejected from the nozzle outlet through plain-orifice with diameter of 3 mm.

2.3 Boundary conditions

Model boundary conditions are as follows: the inlet is set as the pressure boundary condition, outlet boundary condition is set as the pressure boundary, wall boundary condition is set as no-slip walls. The simulation boundary conditions: inlet pressure is set as 10.2 MPa, the outlet pressure is set as 0 MPa, the mass fraction of noncondensable gas is set as 1×10^{-6}. The properties of the working fluid water are based on the room temperature of the fluid, i.e., 20 °C.

2.4 Validation of simulation models

The two-dimensional computational domain is divided by quadrilateral structure grid under the column coordinate system because of geometrical symmetry. The cavitation model implemented in fluent models the formation of bubbles when the local pressure becomes less than the vaporization pressure. A high-density mesh was generated for the orifice. To eliminate the impact of the grid further, the grid-independent test was implemented. The final structured mesh consists of 22,440 quadrilateral elements. Here are results by verifying grid-independent.

Table 2 shows the error between simulation and literature values about coefficient of discharge. From table 2, we can see the values of simulation and literature about coefficient of discharge have the same trend with the change of r/d, and the error is less than 1 %, which proves that the calculated results are accurate and reliable.

Table 2: The error between simulation and literature values about coefficient of discharge

r/d	C_d (simulation)	C_d (Schmidt et al., 1997)	The error (%)
0.030	0.668	0.673	0.743
0.050	0.702	0.707	0.707
0.070	0.730	0.728	0.275
0.090	0.746	0.751	0.666
0.110	0.765	0.772	0.907
0.130	0.782	0.783	0.128
0.150	0.800	0.799	0.125
0.170	0.8125	0.810	0.309
0.190	0.819	0.821	0.244

3. Establishment of new judgment model

3.1 Flaws of judgment rules

From Figure 2, we can see coefficient of discharge versus the injection pressure under the different r/d. The change of three stages also shows the flow pattern in the nozzle from cavitation to hydraulic flip. Therefore, the coefficient of discharge can also be used as an important indicator of the flow pattern in the nozzle, which is the reason why many researchers have used the coefficient of discharge directly as a criterion to determine the flow pattern of the nozzle.

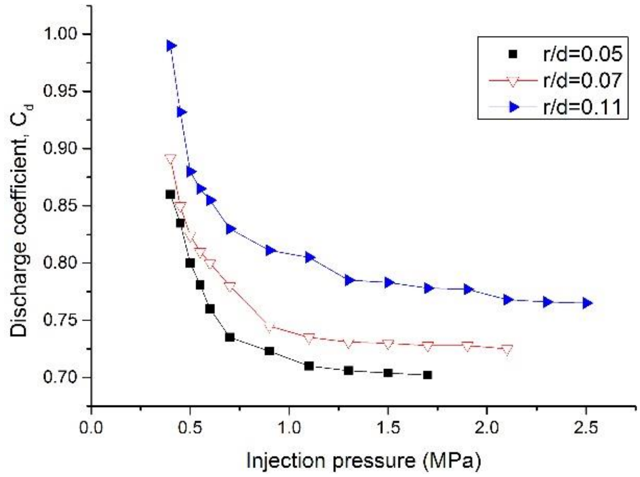

Figure 2: Coefficient of discharge versus the injection pressure

It is simple to use coefficient of discharge as an indicator to characterize the change of flow pattern, but it cannot meet the requirement of the high precision control of the industry. If we want to strengthen atomization by increasing the inlet pressure, at this time we need to determine whether the increase in pressure will lead to the flow regime inside plain-orifice nozzle from cavitation to hydraulic flip.

3.2 Ideas of judgment model establishment

From the existing research, we find the two key factors of affecting hydraulic flip are r/d and L/d. When the value of r/d is greater than a certain value, hydraulic flip does not occur, and the same happens to the L/d. Thus we can change the value of L/d to prevent the occurrence of hydraulic flip, when the value of r/d is less than the certain value. We can see that there is an exponential function between $(L/d)_{min}$ and r/d from the literature (FLUENT V6.2 Documentation, 2003). So we hope to get a more accurate exponential function to express the relationship between r/d, $(L/d)_{min}$ and flow pattern.

3.3 Determination of $(L/d)_{min}$

We think that once the cavitation reached the orifice outlet, ambient air flowed upward into the orifice until the cavitation filled space was filled with the downstream air, and referred to this condition as hydraulic flip. Figure 3 and Figure 4 are the velocity vector at the nozzle exit under r/d = 0, L/d = 9.5 and L/d = 10. From Figure 3, we can see backflow occurs at the nozzle exit, which shows the hydraulic flip can occur. From Figure 4, we can see backflow does not occur at the nozzle exit, which shows the hydraulic flip cannot occur. In this paper, the average value of L/d = 9.8 between L/d = 9.5 and L/d = 10 is taken as the value of $(L/d)_{min}$. The other value of $(L/d)_{min}$ under different r/d conditions can be got by the same way.

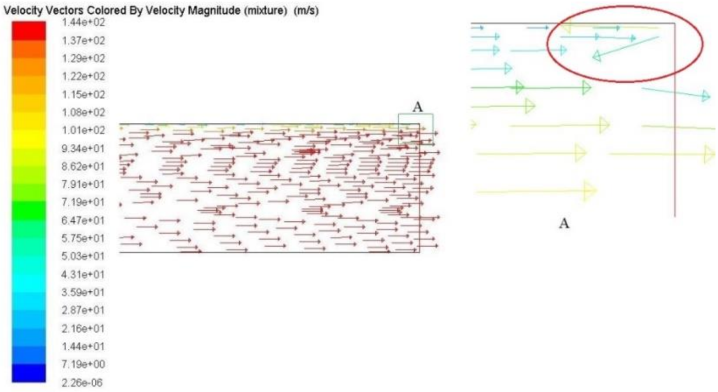

Figure 3: r/d=0、L/d=9.5，the velocity variation at nozzle outlet

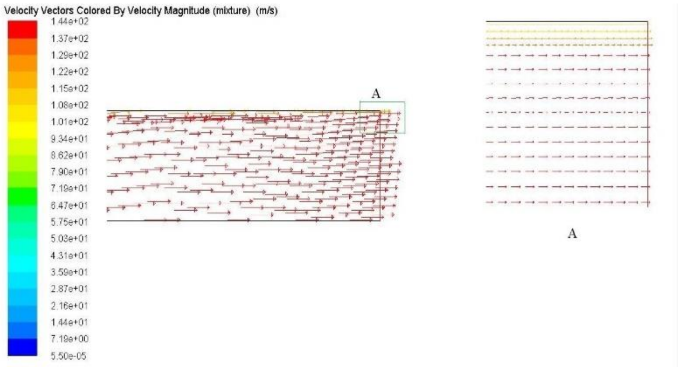

Figure 4: r/d=0、L/d=10，the velocity variation at nozzle outlet

3.4 Regression of new judgment model

The value of $(L/d)_{min}$ corresponding to different r/d was obtained under the condition that the injection pressure was less than 10.2 MPa. The functional relation between r/d, $(L/d)_{min}$ and flow pattern was returned, as shown in Figure 5.

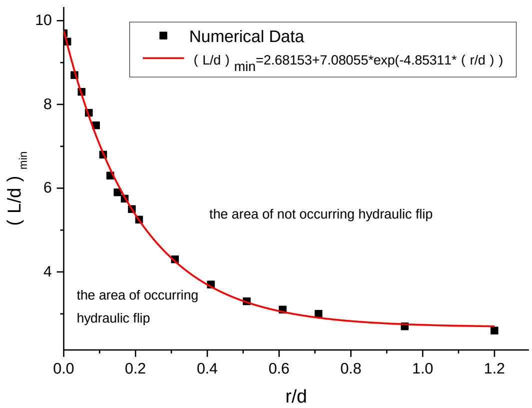

Figure 5: The change of flow with versus L/d and r/d

The functions are as follows:

$$(L/d)_{min} = 2.68153 + 7.08055\,e^{-4.85311(r/d)} \quad (4)$$

From Figure 5, we can see that hydraulic flip cannot occur in the above area of the function line, and the value of $(L/d)_{min}$ decreases with the increase of r/d. The researcher can determine whether the hydraulic flip occurs in the nozzle by function. The reason is that the smaller r/d, the stronger the disturbance of the fluid inside the nozzle, the greater the strength of the cavitation. In order to make the cavitation region cannot reach the nozzle exit and form hydraulic flip, we must control the L/d. In other word, in the case of the hydraulic flip has occurred, when the value of L/d is greater than $(L/d)_{min}$, it is difficult to maintain the stability of axial symmetrical shape. The phenomenon of reattachment occurred. Local cavitation region was recovered.

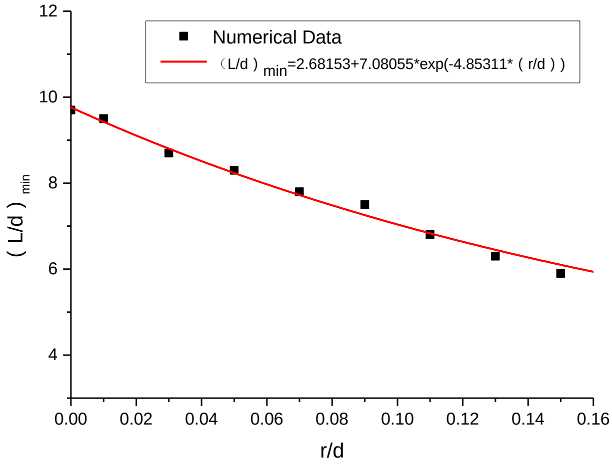

Figure 6: The partially enlarged view of Figure 5

Coefficient of discharge is used as an indicator to characterize the change of flow pattern. If r/d is greater than 0.05, then flip is deemed impossible. From Figure 6, we can see the value of $(L/d)_{min}$ is 8.3 when the value of r/d is 0.05, in other word, when the value of $(L/d)_{min}$ is less than 8.3, hydraulic flip can still occur. From Figure 6, we can obtain when the value of r/d is greater than 0.05, there is still a large operating range, so that the flow regime inside plain-orifice nozzle is hydraulic flip. According to the function of this paper, we can prevent the occurrence of hydraulic flip by changing the geometry of nozzle and improve atomizing effect by increasing pressure.

3.5 Validation of the judgment model

Table 3 shows comparisons of measured by literature, calculated by empirical models (FLUENT V6.3 Documentation, 2003) and the new model data, where the case of occurring hydraulic flip is represented as Y and the case of not occurring hydraulic flip is represented as N. The researcher can accurately judge the hydraulic flip in most cases by the empirical models and new models. However, there are some problem in judgment of hydraulic flip with empirical models, and at this time using the new model to judge hydraulic flip is still consistent with the literature. It shows that the new model is more accurate than the empirical model.

Table 3: Mathematical model validation

Parameter	d(mm)	The case of occurring hydraulic flip			Literature
		Literature	Empirical models	New models	
r/d=0, L/d=4	3	Y	Y	Y	Hiroyasu, 2000
	0.5	Y	Y	Y	Nurick, 1976
	4	Y	Y	Y	Sou et al., 2010
	0.3	Y	Y	Y	Tamaki et al., 2001
r/d=0, L/d=20	0.5	N	Y	N	Nurick, 1976; Ahn et al., 2006
r/d=0, L/d=20	1	N	Y	N	Ahn et al., 2006
r/d=1, L/d=4	3	N	N	N	Hiroyasu, 2000
r/d=1, L/d=20	1	N	N	N	Ahn et al., 2006
	0.5	N	N	N	
r/d=0, L/d=5	0.5	Y	Y	Y	
	1	Y	Y	Y	
r/d=0, L/d=2	4	Y	Y	Y	Sou et al., 2010
r/d=0, L/d=8	4	N	Y	Y	
r/d=0.033, L/d=10	1.52	N	Y	N	Nurick, 1976
r/d=0.044, L/d=10	1.73	N	Y	N	

4. Conclusions

In order to investigate the change of flow pattern in a circular nozzle, a numerical simulation was conducted by using the full cavitation model in the CFD code FLUENT. The conclusions are as follows:

(1) The relationship between the change of flow pattern, r/d and L/d was obtained, and the relation curves between L/d and r/d were plotted, and a new model of judgment of flow pattern was obtained.

(2) At present, there are defects in judgment of flow pattern with empirical model. The model established in this paper is more accurate than the empirical model to judge the change of flow pattern.

(3) The new model is helpful to modify the existing empirical model. In addition, according to the relationship, we can use the method of reasonably adjusting the relationship between the nozzle structure and operating conditions to avoid the generation of hydraulic flip, and to get better atomization effect.

References

Ahn K., Kim J., Yoon Y., 2006, Effects of orifice internal flow on transverse injection into subsonic crossflows: cavitation and hydraulic flip, Atomization and Sprays, 16, 15-34.

Fluent Inc, 2003, Fluent User’s Guide, USA.

Ghassemieh E., Versteeg H., Acar M., 2006, The effect of nozzle geometry on the flow characteristics of small water jets, IMechE, 220(12), 1739-1753.

Hiroyasu H., 2000, Spray breakup mechanism from the hole-type nozzle and its applications, Atomization and Sprays, 10(3), 511-527.

Hong J. G., Ku K. W., Kim S. R. Lee C. W., 2010, Effect of cavitation in circular nozzle and elliptical nozzles on the spray characteristic, Atomization and Sprays, 10(20), 877-886.

Hong J. G., Ku K. W., Lee C. W., 2011, Numerical simulation of the cavitating flow in an elliptical nozzle, J. Atomization Sprays, 21, 237–248.

Ku K. W., Hong J. G., Lee C., 2011, Effect of internal flow structure in circular and elliptical nozzles on spray characteristics, Atomization and Sprays, 21(8), 655-672.

Nurick W. H., 1976, Orifice cavitation and its effect on spray mixing, Journal of Fluids Engineering, 98(4), 681-687.

Payri F., Bermudez V., Payri R., Salvador F. J., 2004, The influence of cavitation on the internal flow and the spray characteristics in diesel injection nozzle, Fuel, 83, 419–431.

Schmidt D. P., Rutland C. J., Corradini M. L., 1997, A Numerical Study of cavitating flow through various nozzle shapes, SAE Tech, Paper Ser no. 971597.

Singhal A. K., Athavale M. M., Huiying L., Jiang Y., 2002, Mathematical basis and validation of the full cavitation model, ASME J. Fluids Eng, 124, 617–624.

Soteriou C., Andrews R. Smith M., 1995, Direct injection diesel sprays and the effect of cavitation and hydraulic flip on atomization, SAE Tech, Paper Ser no. 950080.

Sou A., Hosokawa S., Tomiyama A., 2010, Cavitation in nozzles of plain orifice atomizers with various length-to-diameter ratios, Atomization and Sprays, 20(6), 513-524.

Tamaki N., Shimizu M., Hiroyasu H., 2001, Enhancement of the atomization of a liquid jet by cavitation in a nozzle hole, Atomization and Sprays, 11(2), 125-137.