Numerical study on the effect of a lobed nozzle on the flow characteristics of submerged exhaust

T C Miao, T Du, D Z Wu', L Q Wang

Institute of Process Equipment, College of Chemical and Biological Engineering, Zhejiang University
Zheda Road 38 #, 310027 Hangzhou, Zhejiang, China

E-mail: jjjjjyes@163.com

Abstract. In order to investigate the effecting mechanism of nozzle structure on the flow characteristics of submerged exhaust, the processes of air exhausted from a lobed nozzle and a round nozzle into water have been numerically simulated using realizable $k - \varepsilon$ model under the framework of the volume of fluid (VOF) model. Both the flow structure and the upstream pressure fluctuations are taken into consideration. The calculated results are in good agreement with the experimental results, showing that gas exhausted from the lobed nozzle would flow along the axial direction easier. Flow structure of the gas exhausted from the lobed nozzle is more continuous and smoother. The pressure fluctuations in the upstream pipeline would also be reduced when gas exhausted from the lobed nozzle. The resulting analysis indicates that the lobed structure could deflect water flow into the gas jet. The induced water would be mixed into the gas jet in form of small droplets, making the jet more continuous. As a result, the mixed jet flow would be less obstructed by the surrounding water, and the upstream pressure fluctuation would be reduced. The work in this paper partly explained the effecting mechanism of nozzle structure on the flow characteristics of submerged exhaust. The results are useful in the designing of exhaust nozzles.

1. Introduction

Submerged exhaust can be found in many industry fields, such as metallurgy, chemical engineering, environmental engineering, etc. It can also be used for underwater propulsion. All these applications are based on the understanding of the submerged gas jet, especially the flow characteristics of it [1-6]. Submerged exhaust would produce unsteady turbulent flow which is not well understood. The big differences in physical properties between gas and liquid phases can result in complicated flow structures, which make relative studies remain challenging [7].

1 Dazhuan Wu, Email: wudazhuan@zju.edu.cn

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.

Published under licence by IOP Publishing Ltd
Recent studies show that nozzle structures would efficiently affect the flow structures of submerged exhaust. Linck have found that conical projecting nozzle and corrugated converging diverging nozzle would have significant damping effects on submerged gas flow [8]. Arghode studied the characteristics of submerged exhaust using nozzles with circular, square, triangular and elliptical exit cross-sections. He found that the two phase flow structure and the pressure fluctuation in the upstream pipeline would be effected by these different nozzle structures [9]. Similar studies were also done by Hao who found that both the nozzle size and structure would effect on the gas behavior near the nozzle exit [10, 11]. All these studies indicated that it would be possible to control the flow structure and its associated characteristics by designing different nozzle structures. However, how the nozzle structure would effect on the flow structure is still not clear, relative designs are lack of guidance.

This paper aims to study the effecting mechanism of nozzle structure on the flow characteristics of submerged exhaust using numerical methods. The processes of air exhausted from a lobed nozzle and a round nozzle into water have been numerically simulated using realizable k-ε model under the framework of the volume of fluid (VOF) model. Both the flow structure and the upstream pressure fluctuations are considered. How the nozzle structure would effect on the flow characteristics of submerged exhaust is analyzed.

2. Calculation model

2.1. Numerical method

In order to calculate the gas-liquid two phase flow, 3D N-S equations are used. The control equations are as follows:

\[ \frac{\partial}{\partial t} (\rho_m) + \nabla (\rho_m \vec{v}_m) = 0 \]  

\[ \frac{\partial}{\partial t} (\rho_m \vec{v}_m) + \nabla [\rho_m (\vec{v}_m \cdot \nabla \vec{v}_m) + \rho_m \vec{F} + \nabla (\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k})] = 0 \]  

\[ \frac{\partial (\rho_m \vec{v}_m)}{\partial t} + \nabla (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^T)] + \rho_m \vec{F} + \nabla (\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k}) \]  

\[ \frac{\partial (\rho_m \vec{v}_m \vec{v}_m)}{\partial t} + \nabla (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^T)] + \rho_m \vec{F} + \nabla (\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k}) \]  

\[ \frac{\partial (\rho_m \vec{v}_m \vec{v}_m)}{\partial t} + \nabla (\rho_m \vec{v}_m \vec{v}_m) = -\nabla p + \nabla [\mu_m (\nabla \vec{v}_m + \nabla \vec{v}_m^T)] + \rho_m \vec{F} + \nabla (\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k}) \]  

Where \( \vec{v}_m \) is the average velocity, \( \rho_m \) is the mixed density, \( \rho_k \) is the density of phase k, \( \alpha_k \) is the volume fraction of phase k, \( \vec{v}_m \) is the drift velocity of phase k, \( \alpha_k \) is the mixed viscosity coefficient, \( \mu_m \) is mixed viscosity coefficient, \( \mu_m = \sum_{k=1}^{n} \alpha_k \mu_k \) is the viscous dissipation term.

In order to simulate this situation effectively, a simplified physical model should be used. The following basic assumptions are put forward: 1) The influence of the wall is ignored; 2) As the inlet gas
velocity is 79.6 m/s, the corresponding Maher number is 0.23<0.3, hence the compressibility of the gas medium can be ignored; 3) The mass and heat transfer between phases are ignored.

The turbulence model used in this paper is realizable $k-\varepsilon$ model. The multiphase model used here is VOF (Volume of Fluid) model which can capture the phase interface effectively. The gas medium is air and the liquid medium is water.

2.2. Calculation model parameter and boundary conditions
A schematic of the calculation model is shown in figure.1. The computation region is a cylinder with height of 1000 mm and radius of 600 mm. It was filled with 660 mm depth of water. The direction of gravity is along the axil of the cylinder towards the exhaust nozzle. The exhaust nozzle is placed in the bottom of the cylinder vertically. The diameter of the nozzle inlet is 20mm. Detailed parameters of the lobed nozzle are shown in Fig.1 (b). The trailing edge of the lobed nozzle is formed by six lobe structures. Lobe width and height are 4 mm and 10 mm respectively. The outer and inner penetration angles are both 15°. The mesh generation result of this model is shown in figure.2. The computation region contains 887946 structured grids. We use velocity inlet and pressure outlet as the boundary conditions, the others are set as wall.

![Schematic of the computation model](image1.png)

![Detailed parameters of the lobed nozzle](image2.png)

**Figure.1** Schematic of the computation model
3. Results and analysis

Firstly, the calculating results are compared with the experiment results. As shown in Fig.3, the calculated flow structures near the nozzle exit are in good agreement with the experiment results (DN20 nozzle, the gas flow rate is 90 m$^3$/h). This means the numerical methods we used in this paper is suitable to calculate the submerged exhaust process.

Then we calculated the flow structures of gas exhausted from the round nozzle and the lobed nozzle respectively. As we can see, an obvious stem like structure can be observed when bubble detach from the round nozzle (Fig.4 (a) and (b)). In general, this structure would occur when the axial flow is obstructed. However, similar structure cannot be observed when bubble detach from the lobed nozzle, which means the axial flow via the lobed nozzle is more continuous. Bubbles detached from the round nozzle have smooth
shapes due to the regular cross section shape of the nozzle exit. However, bubbles detached from the lobed nozzle have unstable surfaces which are easily break into small bubbles. Additionally, many small droplets can be seen in the bubbles generated from the lobed nozzle. This is because the lobe trough can guide the surrounding liquid flow towards the gas jet, hence the bubble bottom would be broken up and small droplets would be induced into the gas flow. The small bubbles and droplets exist in the flow field of submerged exhaust through the lobed nozzle would promote the momentum exchange between gas and liquid. As a result, gas exhausted from the lobed nozzle would be less obstructed in the axial direction. This could be proofed by Fig.5. As we can see, gas exhausted from the lobed nozzle (Fig.5 (a)) has thinner jet width and would flow a longer distance along the axial direction within 0.33 s than that from the round nozzle (Fig.5 (b)).

![Fig.4. The calculated flow structures at t=0.15 s, (a), (b) round nozzle, (c), (d) lobed nozzle.](image)

![Fig.5. The calculated flow structures at t=0.33 s, (a) round nozzle, (b) lobed nozzle.](image)

The pressure fluctuations in the exhaust nozzle are measured at point of (0, 0, 30). The measurement results are shown in Fig.6. It can be seen that the pressure fluctuation of gas exhausted from the round nozzle has larger amplitude and higher frequency (Fig.6 (a)). In order to investigate the reason of this phenomenon, the flow structures corresponding to the pressure peaks are also shown in Fig.6. As we can see, the pressure peaks would occur when the gas flow is obstructed in the axial direction. Since the gas exhausted from the lobed nozzle is less obstructed, the corresponding pressure fluctuations are weaker.
Fig.6. The upstream pressure fluctuations and the corresponding flow structures, (a) pressure fluctuations, (b) flow structure corresponding to peak A, (c) flow structure corresponding to peak B

4. Conclusions
In this paper, the submerged exhaust through a round nozzle and a lobed nozzle are numerically calculated using realizable $k-\varepsilon$ model and VOF model. The calculating results coincide well with the experiment results, which means the calculating methods we used here are suitable and reliable. Based on that, the flow structures and pressure fluctuations are analyzed. The main conclusions are as follows:

1) The lobed nozzle would induce small bubbles and droplets into the flow field. This is helpful to make the gas flow more continuous and less obstructed in the axial direction.

2) The pressure peaks in the upstream pipeline are caused by the obstructed gas flow. Since gas exhausted from the lobed nozzle is less obstructed, the pressure fluctuations of it have lower amplitude and frequency.

References
[1] Rue D 2004 Am. Ceram. Soc. Bull. 83 18-20.
[2] Pioro L S, Osnach A M, Pioro I L 2003 Ind. Eng. Chem. Res. 42 6697-6704.
[3] Ozawa, Y., Mori, K. 1986 Trans. Iron Steel Inst. Jpn. 26 291-297.
[4] M. Jamialahmadi, M.R. Zehtaban, H. Muller-Steinhagen, et al. 2001 Chem. Eng. Res. Des. 79 523-532.
[5] Sanada, T., Watanabe, M., Fukano, T., Kariyasaki A. 2005 *Chem. Eng. Sci.* **60** 4886-4900.

[6] S. S. Gulawani, S. S. Deshpande, J. B. Joshi, et al. 2007 *Ind. Eng. Chem. Res.* **46** 3188-3218.

[7] Tang J N, Tseng C C, Wang N F, Shyy W 2011 Flow structures of gaseous jets injected into water for underwater propulsion *AIAA* pp 185.

[8] Linck, Martin B., Ashwani K. Gupta, and Kenneth H. 2009 *J. Propul. Power* **25.2** 522-532.

[9] Arghode, V. K., Gupta, A. K. 2012 *Appl. Energy.* **89** 246-253.

[10] Hao Z R, Wang L Q, Wu D Z, et al. 2011 *J. Eng. Thermophys.* **32** 949-952.

[11] Hao Z R, Zhou Z H 2012 *App. Mech. Mater.* **204** 4620-4623.