Research on airflow uniformity of marine selective catalytic reduction reverse blow system

Man Wang\(^1\), Hongpeng Zhang\(^1\) \& Wenke Gong\(^1\)
\(^1\)Dalian Maritime University, Dalian, People’s Republic of China
\(^\text{E-mail: zhppeter@163.com}\)

Abstract: In order to improve the working performance of the ceramic catalyst filter, a model of nozzle blowback was designed to change the nozzle diameter. The blowback tube blowback model with variable diameter and the blowback blowback model with equal diameter were used for backflush simulation analysis. Also the influence of the sum of the injection tube inlet velocity, nozzle length, and nozzle cross-sectional area on the uniformity of backflush was studied. Through simulation analysis, it can be obtained that the outlet flow of nozzles with equal diameters and blowback models increases gradually from front to back; the variable-diameter blowback model can make the nozzle outlet flow uniform, but the nozzles at both ends still deviate from the average flow; continuing to change the nozzle diameter. The nozzle outlet flow can continue to be uniform; different boundary conditions and sizes can affect the uniformity of the backflush: the smaller the inlet velocity of the injection tube, the larger the nozzle length, the smaller the sum of the nozzle areas, and the more uniform the nozzle outlet flow.

1 Introduction

With the economic development, the process of economic globalisation has been accelerating, and shipping has gradually become the main tool for world trade exchanges. However, the pollution caused by marine diesel engine exhaust gas has gradually attracted the attention of IMO and other countries. Various countries are committed to the research of ship energy-saving and emission reduction devices. The selective catalytic reduction (SCR) device has a significant denitrification effect. Every country is developing this technology \([1, 2]\). However, during the process of using the SCR device, there will be solid particles attached to the surface of the ceramic catalyst-filter and clogging the filter channel. The conventional solution is to use the equal-diameter nozzle for back blowing, but this will cause the backflush to the nozzle outlet flow is not uniform, so that the efficiency of the filter device is reduced, and the life is reduced. In this paper, we study the uniformity of the blowback flow for the backflush process. First, a theoretical calculation was performed to establish a blowback blowback model containing 16 nozzles, and the diameter of each nozzle was calculated \([3–5]\). Next, a physical model was established based on the mathematical model of the injection pipe, and the injection pipe of the equal-diameter nozzle and the injection pipe of the variable-diameter nozzle were simulated to compare the uniformity of the outlet flow of each nozzle and optimise it \([4–7]\). Finally, the influence of the boundary conditions and related dimensions on the uniformity of the nozzle outlet flow was studied.

2 Backflush uniformity mathematical model

Backflush devices mainly include blowpipes, nozzles, and ejectors. When the throttle valve receives the injection signal, it will open automatically. The high-pressure air will enter the injection pipe through the throttle valve and various pipe fittings. It will be injected through the nozzle. The ejected air will suck the surrounding air into the ejector and be carried in the ejector. The exchange of energy and momentum fully mixed into the filter device for backflush \([8, 9]\).

2.1 Blowpipe

The length of the blowpipe in the backwash ashing device is determined according to the number of rows of nanofibre ceramic filters that need to be blown and cleaned. In the blowpipe, the dynamic pressure of the air is rapidly converted to static pressure and is formed in the blowpipe. In the blowpipe, the dynamic pressure of the air is rapidly converted to static pressure and a maximum pressure is formed in the blowpipe. However, the total cross-sectional area of the nozzle aperture on the injection tube cannot be too large. If the total cross-sectional area is too large, the local dynamic pressure at the two ends of the injection tube will become large, and the air flow emitted from the nozzle will be severely blown. The phenomenon, suggestion \(\sum f/F < 1.00\) \((\sum f\) represents the sum of the cross-sectional areas of all nozzles on the blowing tube. \(F\) represents the cross-sectional area of the blowpipe.)

2.2 Nozzle

The function of the nozzle on the blowing tube is to direct the blowing gas stream to the nano-ceramic filter. Studies have shown that the nozzle size, length, and shape of the nozzle will affect the cleaning effect. There are mainly three types of nozzles used to blow clean the dust: taper, zoom, and straight tube. Its structure is shown in Fig. 1 \([10–12]\). At present, there are mainly straight nozzles \([13]\) used at home and abroad, so the type of nozzle used in this paper is a straight nozzle.

![Fig. 1 The schematic diagram of different kinds of nozzle structure](http://creativecommons.org/licenses/by/3.0/)

(a) Straight nozzle, (b) Zoom nozzle, (c) Tapered nozzle
2.3 Nozzle diameter uniformity calculation

In the course of backflushing, the air flow is fast and short, and it can be considered that there is no heat exchange with the outside world and the whole process is insulated. In order to simplify the backflush process, it is assumed that the gas flow into the injection tube is uniform and the velocity is perpendicular to the cross-section of the injection tube. The inlet pressure of the injection pipe and the drag coefficient along the line remain unchanged, and the air outflow is only due to the static pressure of the pipe.

Let the length of the torch be \( L \). The cross-sectional area of the injection tube is \( A \). The distance between each nozzle is equal. The cross-sectional area of the nozzle is different from the foremost end of the injection tube. This ensures the uniformity of the injected air flow. The nozzles are numbered sequentially from the very end of the injection pipe, and the nozzle at the end is No. 1 nozzle.

Nozzles have a cross-sectional area of:

\[
\hat{A}_{i+1} = \frac{Q_i}{n} \tag{1}
\]

where \( \hat{A}_{i+1} \) is the nozzle cross-sectional area, \( Q_i \) is the flow for the injection pipe inlet, \( n \) is the number of nozzles, \( v_{i+1} \) is the first \( i+1 \) nozzle nozzle outflow speed.

The first nozzle discharge speed is

\[
v_{i+1} = \mu \sqrt{\frac{2p_{i+1}}{\rho}} \tag{2}
\]

where \( \mu \) is the nozzle flow coefficient, this paper assumes \( \mu = 1 \), \( p_{i+1} \) is the blowing pipe in the \( i+1 \) nozzle static air pressure, and \( \rho \) is air density.

The centerline cross-section energy equation of the \( i \) and \( i+1 \) nozzles injection tube is:

\[
p_{i+1} = p_i - \frac{\rho Q_i^2}{2} + \frac{\rho Q_i^2}{2} + \Delta p \tag{3}
\]

where \( \Delta p \) is the resistance along the two sections of the blowpipe, \( p_i \) is the \( i \) of air static pressure at the nozzle, \( \omega_i \) with \( \omega_{i+1} \) is the \( i \) and the \( i+1 \) nozzle injection tube flow rate.

Combining (1) and (2) to obtain the \( i \) nozzle static air pressure

\[
p_i = \frac{\rho v_i^2}{2\mu} = \frac{\rho Q_i^2}{2\mu n^2 \delta_i} \tag{4}
\]

The air flow rate in the injection tube is

\[
\omega_{i+1} = \frac{Q_{i+1}}{A} \tag{5}
\]

\[
\omega_i = \frac{Q_i}{A} \tag{6}
\]

Since the air flow is uniform for each blow pipe, the flow rate of each nozzle is equal and can be obtained

\[
Q_i = \frac{Q}{n} (i+1) \tag{7}
\]

Combining (5)–(9) gives

\[
\omega_{i+1} = \frac{Q (i+1)}{n A} \tag{9}
\]

\[
\omega = \frac{Q i}{n A} \tag{10}
\]

The resistance along the center cross-section of the \( i \) and \( i+1 \) nozzles injection tube is:

\[
\Delta p = \int_{b}^{p} \left( \frac{\rho Q_i^2}{2} - \frac{\rho Q_{i+1}^2}{2} + \Delta \frac{\lambda}{n} \right) \frac{dx}{2} \tag{11}
\]

where \( d \) is the diameter of the blowpipe, \( \lambda \) is the coefficient of resistance along the way, \( l \) is the distance between the first nozzle and the last nozzle.

Solve (2) through (11) to obtain the area of each nozzle

\[
\hat{A}_{i+1} = \frac{1}{\sqrt{\left(\frac{(\mu^2 A)^2}{(i+1)^2} - \frac{\lambda}{\mu} \frac{\lambda}{n} \frac{\lambda}{l^2} \right)}} \tag{12}
\]

The above formula is organised into the following formula:

\[
\frac{1}{\hat{A}_{i+1}} = \frac{1}{\hat{A}_{i}} - \frac{\mu^2}{\mu^2 A^2} \left( \frac{1}{(i+1)^2} - \frac{1}{i^2} - \frac{\lambda}{\lambda} \frac{\lambda}{n} \right) \tag{13}
\]

with \( i = 1, 2, 3, 4, \ldots \), substitute into (13) to get equations, simultaneous equations solved:

\[
\hat{A} = \frac{1}{\sqrt{\left(\frac{(\mu^2 A)^2}{(i+1)^2} - \frac{\lambda}{\mu} \frac{\lambda}{n} \frac{\lambda}{l^2} \right)}} \tag{14}
\]

The coefficient of resistance along the turbulent flow of a smooth tube is

\[
\lambda \approx \frac{0.316}{Re^{0.5}} \tag{15}
\]

Established by (14) Relationship, as long as the area of the first nozzle is given, the area of other nozzles will be obtained. Just make sure that the sum of the area of all nozzles is smaller than the cross-sectional area of the injection tube. Owing to the small drag coefficient along the path, this paper takes.

This paper selects the 100 mm diameter and 2000 mm total length of the injection tube. The length of the nozzle at the end of the injection tube is 22 mm, the length of the nozzle is 50 mm, the distance between the nozzles is 100 mm, and the distance between the endmost nozzle and the end of the injection tube is 100 mm. The distance between the head end nozzle and the head of the injection tube is 400 mm. The specific dimensions are shown in Fig. 2.

After theoretical calculations, the diameter of each nozzle is shown in Table 1 and \( \sum f / F = 89.6\% \).
### Table 1 Diameters of the nozzles

| Nozzle number | Nozzle cross-sectional area, mm$^2$ | Nozzle diameter, mm |
|---------------|-------------------------------------|---------------------|
| 1             | 379.94                              | 22.00               |
| 2             | 381.28                              | 22.04               |
| 3             | 383.35                              | 22.10               |
| 4             | 386.80                              | 22.20               |
| 5             | 391.10                              | 22.32               |
| 6             | 396.54                              | 22.48               |
| 7             | 403.29                              | 22.67               |
| 8             | 411.52                              | 22.90               |
| 9             | 421.48                              | 23.17               |
| 10            | 433.52                              | 23.50               |
| 11            | 448.11                              | 23.89               |
| 12            | 465.91                              | 24.36               |
| 13            | 487.89                              | 24.93               |
| 14            | 515.51                              | 25.63               |
| 15            | 551.12                              | 26.50               |
| 16            | 598.76                              | 27.60               |

#### 3 Spray pipe simulation model

##### 3.1 $k-\varepsilon$ turbulence model

The $k-\varepsilon$ turbulent viscosity coefficient of the turbulence model is solved by solving the turbulent pulsation ($k$ Equation) and turbulence dissipation rate ($\varepsilon$ Equation) to solve [14, 15]. (see (16) and (17)) . In the formula, $G_0$ is the turbulent kinetic energy caused by the laminar velocity gradient; $k$ is the generated items; $G_0$ is the buoyancy-induced turbulent kinetic energy. Generated items; $Y_M$ is the fluctuation caused by the diffusion phenomenon in the compressible turbulent fluid; $C_{ii}$, $C_{ij}$ are constants; $\sigma_i$ is the Mobility energy; $k$ is the corresponding Prandtl number; $\sigma_i$ is the Dissipation rate; $\varepsilon$ is the corresponding Prandtl number; $C_{ii}$ is a buoyancy-related parameter. If the fluid moves in a direction parallel to the direction of gravity, $C_{ii} = 1$. If the fluid moves in a direction perpendicular to the direction of gravity, $C_{ii} = 0$. $S_1$, $S_2$ are the customise of the item for the user.

##### 3.2 Grid division and boundary conditions

A three-dimensional simulation model including a blowpipe and a nozzle is established, as shown in Fig. 3. The grid number of this model is 254204, and the convergence standard is set to $10^{-3}$ in the inlet of the injection pipe is set to the speed inlet. The walls of the injection pipe and the 16 nozzles are set to the wall. The outlet of the nozzle is set to the pressure outlets in order to form the extreme end of the injection pipe.

Since the backflush process is short, the blowback gas flow through the blowpipe and the nozzle is very fast, the pressure changes rapidly, and the flow field of the entire backflush process is very complex. Therefore, the following assumptions are made for the simulation model: the heat exchange process is neglected; The gas in the field is considered to be an ideal gas and is an incompressible gas; the pipe wall has no resistance to the fluid.

#### 3.3 Setting of gas parameters

Since the pressure of marine compressed air is 3 MPa, the pressure at the inlet to the injection tube is reduced through the friction of various valve parts and pipelines, and the pressure at the inlet of the injection tube of the SCR system is set to 0.5 MPa, assuming a temperature of 27°C. Calculated by $\rho = 6.02$. The temperature is 300 K. At this temperature, the air viscosity named $\mu$ is $1.74 \times 10^{-5}$ Pa·s. The turbulence intensity named $I$ is 0.027.

#### 4 Simulation analysis and optimisation of equal-diameter and variable-diameter nozzles

##### 4.1 Simulation analysis of equal-diameter and Variable-diameter nozzles

An equal-diameter nozzle model was established and the nozzle diameter was set to 22 mm. The boundary conditions are: the inlet flow rate of the injection tube is 50 m/s, the inlet pressure is 0.5 MPa, the temperature is set to 27°C, and the air viscosity is the density is $6.02 \text{ kg/m}^3$. Turbulence intensity is 0.027. The simulation results are shown in Fig. 4.

The size of the flow uniformity can be expressed as a variance

$$
\sigma^2 = \frac{1}{n} \sum_{i=1}^{n} \left( \frac{Q_i - \bar{Q}}{Q} \right)^2
$$

(18)

By (17), the variance of the flow rate of the nozzle with equal diameter is $8 \times 10^{-4}$. The variance of the flow rate of the variable-diameter nozzle is $7.48 \times 10^{-5}$. Variable-diameter nozzles can greatly increase the uniformity of nozzle outlet flow. However, the line chart can also be seen, the flow rate of each nozzle in the middle part is relatively stable, and the nozzle flow at the front and rear ends is very unstable, so it is necessary to optimise it and continue to change the diameter of the nozzle outlet.
4.2 Optimisation simulation of Variable diameter nozzle model

Assuming equal flow for each nozzle, we get

\[
\bar{Q} = \frac{Q}{16}
\]

(19)

\[
K_i = \frac{Q_i}{Q}(i = 1, 2, ..., 16)
\]

(20)

In the formula, \(Q\) is the outlet flow of 16 nozzles, \(\bar{Q}\) is the 16-nozzle flow and the average value, \(Q_i\) is the actual flow rate for each nozzle. \(K_i\) is the ratio of the actual flow rate of each nozzle outlet to the average flow rate of 16 nozzles.

\[
K_iA_i = A_0
\]

(21)

The exit area of the original nozzle is

\[
A_i = \frac{\pi D_i^2}{4}
\]

(22)

The nozzle outlet area after correction is

\[
A_i = \frac{A_0}{K_i}
\]

(23)

The above formula can be corrected to get the corrected nozzle diameter

\[
D_i = \sqrt{\frac{1}{K_i}D_0}
\]

(24)

The model of the injection tube was established, given the same boundary conditions: the inlet velocity of the injection tube was 50 m/s, the inlet pressure was 500,000 Pa, the temperature was 27°C, and the air viscosity was \(\mu = 1.74 \times 10^{-5}\) Pa·s; The density is 6.02 kg/m³. According to (22) and (23), a new injection model is established, but considering that only the nozzle flow rate at both the front and rear ends varies greatly, in order to simplify the process, only the No. 1 and No. 2 nozzles and 13, 14, and 15 are needed. Nozzle No. 16 can be optimised. The result as shown in Fig. 5.

From the flow table and the line chart, we can see that the optimised nozzle outlet flow is more uniform, and the variance of each nozzle outlet after optimisation is calculated as \(T = 1.28 \times 10^{-5}\). The flow uniformity has greatly improved.

5 Influence of different boundary conditions and different sizes on spray uniformity

5.1 Effect of nozzle speed on injection uniformity

The inlet speeds of the blow ports were set to 50, 100, 150, 200, and 250 m/s, respectively. They were represented by State 1, State 2, State 5, and the turbulence intensity was 0.0265, 0.0244, 0.0223, 0.0212, 0.0204, other parameters and boundary conditions remain unchanged.

Calculate the variance of each nozzle flow from the data in table 7.48 \times 10^{-5}, 2.65 \times 10^{-4}, 7.1 \times 10^{-4}, 1.26 \times 10^{-3}, 1.94 \times 10^{-3}. Therefore, it can be concluded that under the same pipe diameter, the greater the inlet speed, the more uneven the nozzle outlet flow, the lower the inlet speed, and the more uniform the nozzle outlet flow. The line chart is shown in Fig. 6.

5.2 Effect of nozzle length on blowing uniformity

In this paper, nozzles with nozzle lengths of 20, 30, 40, and 50 mm are selected and represented by Model A, Model B, Model C, and Model D, respectively. Simulations are performed. The speed of the inlet is set to 50 m/s. Other parameters and boundaries The
With the increasingly stringent restrictions on NO X emissions from the MARPOL Convention, and the establishment of ship emission control zones at home and abroad, the standards for the control of ship pollutant emissions will become more and more stringent. Both at home and abroad are devoted to the study of how to reduce ship emissions. And improve the efficiency of the equipment. In this paper, a 16-nozzle injection-tube injection model is established. The nozzle discharge flow rate is used as the standard to study the uniformity of the injection, and the optimisation of the injection nozzle is performed to analyse different boundary conditions and different sizes. The impact of sex. The following conclusions are drawn: The outlet flow rate of nozzles with equal-diameter blowback nozzles is very uneven. By changing the outlet area of the nozzles, the uniformity of the injection can be improved. However, the outlet flow rate of the front and rear nozzles is still deviated from the average flow rate. The uniformity of the injection can be further improved. Different boundary conditions and sizes have a certain influence on the uniformity of the blowback flow: the greater the inlet velocity of the injection tube, the worse the injection uniformity; the greater the nozzle length, the better the injection uniformity; the sum of the nozzle area and the spray The smaller the blowpipe area ratio, the better the injection uniformity.

6 Conclusion

With the increasingly stringent restrictions on NO X emissions from the MARPOL Convention, and the establishment of ship emission control zones at home and abroad, the standards for the...