Fluid Dynamics Research for Nozzle Flow with the Computational Fluid Dynamics

Xiangyuan Kong 1, a, †, Dongchi Wang 2, b, †

1 College of Engineering, Ohio State University, Columbus, 43210, USA
2 Northwest A&F University, College of Mechanical and Electronic Engineering, Yangling, Shaanxi712100, China

*Corresponding author’s e-mail: akong.434@buckeyemail.osu.edu
bwdc200504@nwafu.edu.cn

†These authors contributed equally.

Abstract. Nozzle is a kind of fluid equipment with a wide range of applications. Research on the internal flow of the nozzle is very important for improving the performance of the nozzle or improving the design of the nozzle. In this study, the flow conditions in the convergent-divergent nozzle, as well as the convergent or divergent nozzles are solved and explored by numerical simulations based on Euler equations. It is found that the fluid Mach number is affected by many factors. With the increase of the density, temperature, and pressure, the fluid Mach number will decrease. In the study of the nozzle shape, a discovery for the nozzle flow rate is that the change of the physical parameters of the fluid will fluctuate due to the increase of the cross-sectional area, and specifically it will slow down this change.

1. Introduction

To understand and investigate the mechanism of various fluid related phenomenon, some basic equations and methods were proposed, such as the Bernoulli equation, Euler method, and Lagrange method, which leads to the establishment of the system of classical fluid dynamics. As for the computational fluid dynamics (CFD), it is an alternative method for modern application and research on fluid dynamics. The birth of the CFD has a strong relationship with the development of computer science. With the help of the CFD, the numerical simulations can be performed during the research. Since the nozzle can accelerate both the liquid and gas, it is widely used in many areas. For instance, the nozzle of a rocket or engine can be redesigned to accelerate the gas according to its principle, so that the speed of the vehicle can be increased significantly.

As a device that plays a key role in many industries, the nozzle has been studied by many researchers, and these studies are related to many different characteristics of the nozzle. In the investigation on the nozzle, if the optimal Mach number is determined, the research on the nozzle often focuses on its shape, which is mainly reflected in the size of the opening [1]. Cetin et al. used Large-eddy simulations to study the turbulence generated by the nozzle of a helicopter engine when the Reynolds number and Mach number were constant and investigated the influences of nozzle shapes [2]. In research for the nozzles on the aircraft, how to determine and calculate the boundary conditions and calculate them is a very important issue in the CFD process [3]. Besides, in the numerical analysis of nozzle flow, the conservation equations for energy, momentum, and mass are
very important. Some studies use Reynolds-averaged Navier-Stokes (RANS) equations to investigate fluid changes over time [3]. For a constant nozzle flow, using numerical analysis with the one-dimension Euler equation in the calculation of the transonic point will be troublesome due to the errors, so that the transonic point cannot be reached. In addition, the speed will jump and become supersonic. To solve this problem, the equations, other than the momentum equation in Euler’s equation, can be written in a conservative form [4]. Moreover, there are PIV measurements and numerical simulation for studying the structure of a single-phase gas flow from a pneumatic nozzle that is used for liquid fuel spraying, as well as the Reynolds stress model (RSM) in the stationary axisymmetric formulation and the DES method used to model turbulence [5]. Furthermore, with the description of numerical simulation and CFD approach, the effects on the geometric parameters of the nozzle on the flow and cavitation characteristics within the nozzle have been numerically investigated by three-dimensional simulation, with a commercial product of six-hole injector being adopted as the basic physical model [6]. Besides, numerical schemes for quasi-1D steady nozzle flows confirm the capacity of these methods to construct well-balanced schemes. The mentioned strategy will be used to extend the flux limiter technique, as well as the Harten, Lax and van Leer Riemann solver to the quasi-1D flow in ducts of variable cross-section [7].

In this work, the state of the fluid in the nozzle will be obtained through solving Euler's equation. Moreover, the influence of the inlet Mach number, density, and nozzle cross-section on the performance of the nozzle will be investigated.

2. Method

Please follow these instructions as carefully as possible so all articles within a conference have the same style to the title page. This paragraph follows a section title so it should not be indented [2-3].

2.1. Euler equation of nozzle

In this paper, Euler's equation is used to study the various parameters of the fluid in the nozzle. The equation of the two-dimensional Euler equation is as follows:

\[
\begin{align*}
\frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + \rho \frac{\partial u}{\partial x} &= 0 \\
\frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + \rho \frac{\partial u}{\partial x} &= 0 \\
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + \rho \frac{\partial u}{\partial x} &= 0
\end{align*}
\]

(1)

where \( \rho, p, u, c \) denote the fluid density, fluid pressure, fluid velocity, speed of sound, respectively. For the convenience of expressing equations in code, the Euler equation is reshaped into the following matrix form:

\[
\begin{align*}
\frac{\partial}{\partial t} \begin{pmatrix} \rho \\ u \\ p \end{pmatrix} + \begin{pmatrix} u & \rho & 0 \\ 0 & u & 1 \\ 0 & \rho & u \end{pmatrix} \begin{pmatrix} \rho \\\ u \\ p \end{pmatrix} = 0
\end{align*}
\]

(2)

By introducing parameters \( U \) and \( F \), the equation can be simplified as:

\[
\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} = 0
\]

(3)

To eliminate the parameters, the equation is introduced as:
\[
\begin{align*}
  p &= (\gamma - 1) \rho c_e \\
  c_e^2 &= \gamma \frac{p}{\rho} \\
  E &= \epsilon + \frac{1}{2} \rho u^2
\end{align*}
\] (4)

where \(\gamma, \epsilon, E\) denote the ratio of specific heat capacity at constant pressure to specific heat capacity at constant volume, the internal energy of the fluid, the total energy of the fluid, respectively.

Incorporating Eq. 4 into Eq. 2 and Eq. 3, the matrix expressions of \(U\) and \(F\) can be obtained:

\[
\begin{align*}
  U &= \begin{pmatrix} \rho \\ \rho u \\ \rho E \end{pmatrix} = \begin{pmatrix} \rho \\ m \\ \epsilon \end{pmatrix} \\
  F &= \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho u H \end{pmatrix} = \begin{pmatrix} m^2 \\ \rho \\ m(\epsilon + p) \end{pmatrix}
\end{align*}
\] (5)

Based on the two-dimensional Euler equation and introducing the function of nozzle area, the Euler equation of the fluid in the nozzle can be calculated:

\[
\begin{align*}
  \frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + \rho \frac{\partial u}{\partial x} &= -\frac{\rho u}{S} \frac{dS}{dx} \\
  \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + \frac{1}{\rho} \frac{\partial p}{\partial x} &= 0 \\
  \frac{\partial p}{\partial t} + u \frac{\partial p}{\partial x} + \rho \epsilon &= -\frac{\rho u c_e^2}{S} \frac{dS}{dx}
\end{align*}
\] (6)

By simplifying the Eq. 7, we can obtain:

\[
\begin{align*}
  \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{S'}{S} (\rho u) &= 0 \\
  \frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2 + u)}{\partial x} + \frac{S'}{S} (\rho u^2) &= 0 \\
  \frac{\partial E}{\partial t} + \frac{\partial u(\epsilon + p)}{\partial x} + \frac{S'}{S} (u(\epsilon + p)) &= 0
\end{align*}
\] (7)

Introducing the parameter \(G\), the equation can be simplified:

\[
\begin{align*}
  \frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{S'}{S} G &= 0
\end{align*}
\] (8)

Incorporating Eq. 4 into Eq. 8 and Eq. 9, we can get the matrix expressions of \(U\), \(F\) and \(G\):
2.2. Numerical solution to the Euler equation

The idea of CFD is to use programming to simulate actual fluid problems. However, since programming cannot express the partial differential form of physical quantities, we differentiate physical quantities, that is, carry out grid differentiation. Since the central difference has the advantages of easy programming, less storage space, simple boundary conditions, and small errors, we choose to perform the central difference on Euler's equation. Use \( j \) and \( n \) to denote the current position and time, respectively, and use \( j+1 \) and \( j-1 \) to denote the physical state of the front and back positions. Similarly, use \( n+1 \) and \( n-1 \) to denote the physical state of the front and back moments. The one-dimensional Euler equation is subjected to central difference:

\[
\rho \frac{\partial U}{\partial t} + \rho u \frac{\partial U}{\partial x} + \frac{\partial F}{\partial x} = 0
\]

\[
\frac{1}{\Delta t} \left[ U_{j+1}^{n+1} - \frac{1}{2} \left( U_{j+1}^{n} + U_{j}^{n} \right) \right] + \frac{1}{2\Delta x} \left( F_{j+1}^{n} - F_{j}^{n} \right) = 0
\]  

In the same way, we can obtain the difference format of the nozzle flow Euler equation by subtracting the center of Eq. 9:

\[
\frac{1}{\Delta t} \left[ U_{j}^{n+1} - \frac{1}{2} \left( U_{j+1}^{n} + U_{j-1}^{n} \right) \right] + \frac{1}{2\Delta x} \left( F_{j+1}^{n} - F_{j}^{n} \right) + \frac{S}{\Delta x} G_{j}^{n} = 0
\]  

2.3. Steps for the solving of discrete equations

Firstly, we define the shape of the nozzle; second, the initial boundary condition is specified to the nozzle; next, we use Euler’s equation to express the nozzle flow. Then, the central difference scheme is employed to discrete the partial differential equations to obtain the discrete equation and solve it iteratively. The next step is to output the results after running the code and perform an error analysis to improve the code. Finally, we get the output image, and the flow law of the nozzle is compared and analysed.

3. Results and discussion

3.1. Baseline nozzle shape and its flow

The shape of the nozzle is narrow in the middle and wide on both sides. It will greatly affect the experimental results, so considering the convenience and practicability of the research, we choose the hyperbolic nozzle as the basic research object:
\[ y = 2 + 2.2 \times (x - 1.5)^2 \]  \hspace{1cm} (12)

Since the range of our research is \( x \in [0, 3] \), we select a hyperbolic nozzle with a symmetry equal to \( x = 1.5 \).

Fig. 1 Shape of the basic nozzle

Fig. 2(a)-(d) show the flow quantities in different position of the nozzle. In those figures, a point is used to mark the location of the throat. As seen in Fig. 2(a)-(d), within the scope of this study, when the fluid flows from the inlet to the throat and then to the outlet, the Mach number gradually increases, and the rate of change at the throat reaches the maximum; on the contrary, pressure, temperature, and density gradually decrease, but the rate of these factors also reaches the maximum at the throat.

Fig. 2 Flow quantities in different position of the nozzle with \( y = 2 + 2.2 \times (x - 1.5)^2 \)
3.2. The influence of nozzle cross-section profile on the flow
In this subsection, the influence of nozzle cross-section profile on the flow will be discussed. Two nozzles with following shapes are investigated additionally:

\[
\begin{align*}
y &= 4 + 2.2 \cdot (x - 1.5)^2 \\
y &= 6 + 2.2 \cdot (x - 1.5)^2
\end{align*}
\] (13)

The shapes of the two additional nozzles are presented in Fig. 3(a) and (b). It can be noted that the cross section is wider compared with the baseline nozzle shown in Fig. 1.

Comparing Fig. 1(a)-(d) with Fig. 4(a)-(d), we can find that when we choose a wider nozzle for the experiment, the difference of inlet and outlet Mach number, fluid pressure, fluid temperature, and fluid density will all become smaller. At the same time, the changing trend of fluid Mach number, fluid pressure, fluid temperature, and fluid density slow down.
The change trend of the dimensionless density and temperature of the fluid

The dimensionless density and temperature of the fluid inside the baseline nozzle are presented in Fig. 6 to have a further discussion about the mechanism of the trends of the flow quantities and potential usage of the observations.

3.3. The change trend of the dimensionless density and temperature of the fluid

When the fluid flows from the inlet to the throat, as the cross-sectional area of the nozzle decreases, the molecular distance in the fluid is squeezed, and the volume of the fluid becomes smaller, so the density of the fluid increases. In the process of flowing from the throat to the outlet, the cross-sectional area of the nozzle increases, the fluid molecules disperse at a uniform speed, the fluid volume becomes larger, and the fluid density becomes smaller. When the fluid flows from the inlet to the throat, as the
cross-sectional area of the nozzle decreases, the molecular distance in the fluid is squeezed, and the molecular thermal motion becomes violent, so the fluid temperature rises. In the process of flowing from the throat to the outlet, the cross-sectional area of the nozzle increases, the fluid molecules disperse at a uniform speed, the molecular energy is dissipated, and the fluid temperature becomes lower.

By this study, the influence of the nozzle shape on the Mach number, pressure, temperature, and density of the fluid can be obtained. Therefore, in the actual manufacturing process, we can select the most suitable nozzle shape according to the actual situation to meet the actual needs. In addition, this article has also studied the changing trend of fluid density and temperature in the nozzle, and we can obtain the fluid state we need by controlling the position of the nozzle. For example, if we need fluid with high temperature, high density, and high energy, we can collect the fluid at the throat for further work.

4. Conclusions and future work
In this study, we used a numerical simulation method based on Euler's equation to solve the flow in the convergent-divergent nozzle. In the baseline of nozzle flow simulation, we found that the density, temperature, and pressure of the fluid gradually increased. At the same time, the Mach number of the fluid gradually decreased. After studying the influence of the nozzle shape on the nozzle flow, we found that as the cross-sectional area increased, the physical parameters of the fluid changed slowly. Our current research is based on Euler's equation. In the future, we may use numerical simulation based on the NS equation to study the flow inside the nozzle more accurately.

References
[1] Meena, L., Niranjan, M. S., Aman, Gautam, Gagandeep, Kumar, G., & Zunaid, M. (2021). Numerical study of convergent-divergent nozzle at different throat diameters and divergence angles. Materials Today: Proceedings, 1-8.
[2] Cetin, M. O., Pauz, V., Meinke, M., & Schröder, W. (2016). Computational analysis of nozzle geometry variations for subsonic turbulent jets. Computers and Fluids, 136: 467–484.
[3] Arif, I., Masud, J., & Shah, I. (2018). Computational Analysis of Integrated Engine Exhaust Nozzle on a Supersonic Fighter Aircraft. Journal of Applied Fluid Mechanics, 11(6): 1511–1520.
[4] Colonna, G., Tuttafesta, M., & Giordano, D. (2001). Numerical methods to solve Euler equations in one-dimensional steady nozzle flow. Computer Physics Communications, 138(3): 213–221.
[5] Sun, Z.-Y., Li, G.-X., Chen, C., Yu, Y.-S., & Gao, G.-X. (2015). Numerical investigation on effects of nozzle’s geometric parameters on the flow and the cavitation characteristics within injector’s nozzle for a high-pressure common-rail DI diesel engine. Energy Conversion and Management, 89: 843–861.
[6] Gascón, L., Corberán, J. M., & García-Manrique, J. A. (2021). Numerical schemes for quasi-1D steady nozzle flows. Applied Mathematics and Computation, 400: 1-14.
[7] Alekseenko, S. V., Anufriev, I. S., Dekterev, A. A., Kuznetsov, V. A., Maltsev, L. I., Minakov, A. V., Chernetskiy, M. Y., Shadrin, E. Y., & Sharypov, O. V. (2019). Experimental and numerical investigation of aerodynamics of a pneumatic nozzle for suspension fuel. International Journal of Heat and Fluid Flow, 77: 288–298.