Simulation of nozzle flow based on Euler equations

Kaipeng Chen¹, a, †, Haoze Liu², b, † and Yinghan Tang³, c, †

¹Department of Machinery and Automobile Engineering, South China University of Technology, Guangzhou, Guangdong, 510000, China
²College of Physics, Jilin University, Changchun, Jilin, 130012, China
³Bourns College of Engineering, University of California Riverside, Riverside, CA 92507, US

*Corresponding author’s e-mail: a201830040024@mail.scut.edu.cn,
b1951253260@qq.com, cytang070@ucr.edu
†These authors contributed equally.

Abstract. The nozzle is a widely used device in daily life, such as water fountains to rocket engines. It is important to find out the influence of the position of the nozzle throat for the application or the design of the nozzle. To that end, the finite difference method was employed to solve the 1D Euler equations to obtain the flow inside the nozzle. To implement the method, an in-house python code was developed. The relationship among the velocity, pressure and density in the convergent-divergent nozzle flow was found. It is observed that: the velocity rose quickly along with the nozzle and reached the top before a rapid decrease; pressure remained constant initially, which eventually began to drop; density dropped steadily and had a turning point. Moreover, the influence of the nozzle throat position is investigated thoroughly. It is observed that the position of the nozzle throat influences the velocities at the nozzle exit. The faster the flow reaches the throat, the higher the velocity or Mach number at the exit boundary.

1. Introduction

Computational fluid dynamics (CFD) is the process that uses numerical analysis and data structure to mathematically model, analyse, and solve problems involving fluid flow [1]. It is commonly used in the industry as an analysis tool in the early development phase of a project, replacing physical prototypes, which can be expensive. The method is seen in car and plane simulations and nozzle flow, which is the topic of this research. A nozzle flow is when a gas or a fluid moves through a narrow opening with no change in entropy. It is seen everywhere from daily life, such as water fountains to rocket engines [2]. Nozzles at the end of rocket engines increase the kinetic energy while lowering its pressure and internal energy.

The investigation of the nozzle is mostly about the diesel injectors. Examples are as followed. Balz et al. [3] in-text citation analysed the cavitation of a marine diesel nozzle. They adopted different methods like the volume of fluid-based numerical simulation, experiment, and homogeneous relaxation model to describe the rate at which the instantaneous quality and the mass fraction of vapor in a two-phase mixture will approach its equilibrium value. Guo et al. [4] investigated the cavitation of a diesel nozzle. They utilized the data of the cavitation build of an optical model and used this model to get the transient flow characteristics inside a five-hole injector nozzle. Their purpose was to reduce fuel gas emissions to protect the environment. [5] The rapid reduction of in-nozzle flow rate at the end of diesel
injection events inhibits spray atomization and releases large slow-moving liquid structures into the cylinder. High-speed optical microscopy was used, and it was discovered that spray wetting was more pronounced at the reduced load conditions. Alya Sachin et al. [6] adopted a discrete model to analyse the mass flow distribution asymmetries at various nozzle inclinations. They found that the angle of the nozzle influenced powder catchment efficiency. He et al. utilized a tomographic particle image velocimetry system to directly measure the velocity in the lobed nozzle. [7] The authors fixed the Reynolds number and measured the circular jet and found the dynamics of the three-dimensional jet flow issued from a lobed nozzle. Shi Gang et al. [8] developed a CFD–based method for optimizing vacuum ultraviolet photoreactor performance. They analysed the energy and considered the effects of two important operating parameters—flow rate and radiant exitance. They wanted to use the method of removing micropollutants such as 1,4-dioxane from source water.

In this paper, the Euler equation was used to get the details of the nozzle flow. In detail, the finite difference method was adopted to transform the Euler equation into a discrete equation. Then it is solved iteratively to obtain the flow quantities, like density, pressure, etc. The influence of the nozzle throat position is investigated thoroughly. In section 2, the methods are presented. Results and discussion are given in Section 3. Conclusions are drawn in Section 4.

2. Method

The nozzle flow is obtained by solving the Euler equations via the finite difference method. Euler Equations are based on three basic conservation laws, conservation of mass, momentum, energy, respectively. Implementing these three laws in the fluid can get the Navier-Stokes equations, which are basic in fluid questions. Navier-Stokes equations are used to describe the viscous flow. When we ignore the influence of viscosity and thermal diffusion, we can get the Euler equations. In other words, Euler equations are one of a simple form in Navier-Stokes equations. Most situations in nature can be solved with Euler equations, so it became the most important theoretical basis to conduct related research. The Euler Equations are expressed as:

\[
\frac{\partial}{\partial t} \left( \rho \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial x} \left( \rho u \frac{\partial u}{\partial x} \right) = -\frac{\partial p}{\partial x} + \rho \frac{\partial u}{\partial x} \frac{\partial \rho}{\partial x} - \rho \frac{\partial u}{\partial x} \frac{\partial (\rho u)}{\partial x}
\]

where \( \rho, u, e, p, c \) denote the density, velocity, total energy per unit volume, pressure, and sound speed.

In nozzle flow, Euler Equations must be adapted to the changing area because of the nozzle shape. Then, we can get the Euler equations for the nozzle flow [2]:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} = 0
\]

\[
\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} = -\frac{\partial p}{\partial x} + \rho \frac{\partial u}{\partial x} \frac{\partial \rho}{\partial x}
\]

Because there are four unknown variables \( \rho, u, p, e \), we eliminate \( p \) and \( e \) with equation

\[
p = \rho RT, \quad \frac{\partial p}{\partial x} = R \left( \rho \frac{\partial T}{\partial x} + T \frac{\partial \rho}{\partial x} \right), \quad e = c_v T
\]

where \( R, T, c_v \) are gas constant, temperature and constant volume specific heat coefficient, respectively. Then, Eq. (2) can be expressed as:

\[
\frac{\partial (\rho A)}{\partial t} + \frac{\partial (\rho u A)}{\partial x} = 0
\]

\[
\rho \frac{\partial u}{\partial t} + \rho u \frac{\partial u}{\partial x} = -R \left( \rho \frac{\partial T}{\partial x} + T \frac{\partial \rho}{\partial x} \right)
\]

\[
\rho c_v \frac{\partial T}{\partial t} + \rho u c_v \frac{\partial T}{\partial x} = -\rho RT \left[ \frac{\partial u}{\partial x} - u \frac{\partial (\rho u)}{\partial x} \right]
\]
In simulation of nozzle flow, we prefer to use the dimensionless form. The results of dimensionless form are straightforward because it’s easy to be compared with constant. To that end, the following dimensionless variables are defined:

\[ \rho' = \frac{\rho}{\rho_0}, \quad u' = \frac{u}{u_0}, \quad A' = \frac{A}{A^*}, \quad t' = \frac{t}{L/\alpha_0}, \quad x' = \frac{x}{L}, \quad e' = \frac{e}{e_0}, \quad T' = \frac{T}{T_0} \] 

(5)

where, \( \rho_0, u_0, A^*, L, \alpha_0 = \sqrt{\gamma R T_0} \) denote the density in plenum chamber, velocity in plenum chamber, cross-sectional area of the throat, length of the nozzle, sound speed, the temperature in plenum chamber, respectively. \( A \) is the cross-sectional area of the nozzle, and we assume that it’s Laval nozzle and the \( A(x) \) used in this study is:

\[ A(x) = 1 + 2.2(x - 1.5)^2 \] 

(6)

With the dimensionless variables, Eq. (3) can be transformed as:

\[
\begin{align*}
\frac{\partial \rho'}{\partial t} &= -u' \frac{\partial \rho'}{\partial x} - \rho' \frac{\partial u'}{\partial x} - \rho' u' \frac{\partial (\ln A')}{\partial x}, \\
\frac{\partial u'}{\partial t} &= -u' \frac{\partial u'}{\partial x} - \frac{1}{\gamma} \left( \frac{\partial T'}{\partial x} + \frac{T'}{\rho'} \frac{\partial \rho'}{\partial x} \right), \\
\frac{\partial T'}{\partial t} &= -u' \frac{\partial T'}{\partial x} - \left( \gamma - 1 \right) T' \left( \frac{\partial u'}{\partial x} + \frac{u'}{\rho'} \frac{\partial \rho'}{\partial x} \right)
\end{align*}
\] 

(7)

where, \( R = c_p (\gamma - 1) \). It is difficult for us to get the exact solution, so we use the finite-difference method, a basic method to solve the partial differential equations (PDEs), to get the approximate solution. The MacCormack method is a pre-estimation and correction finite difference method. It is a variant of the Lax-Wendroff method, that is easier to use. As a finite difference method, it has second order accuracy in time and space. It is one of the simplest and easy to code methods for students to learn, while also providing promising results.

**Pre-estimation step:** the forward difference method is employed to calculate the spatial derivative on the right side of the Eq. (7):

\[
\frac{\partial \rho'}{\partial t} \bigg|_i = -u_i \frac{\rho_{i+1} - \rho_i}{\Delta x} - \rho_i \frac{u_{i+1} - u_i}{\Delta x} - \rho_i u_i \frac{\ln A_{i+1} - \ln A_i}{\Delta x},
\]

\[
\frac{\partial u'}{\partial t} \bigg|_i = -u_i \frac{u_{i+1} - u_i}{\Delta x} - \frac{1}{\gamma} \left( \frac{T_{i+1} - T_i}{\Delta x} + \frac{T_i}{\rho_i} \frac{\rho_{i+1} - \rho_i}{\Delta x} \right),
\]

\[
\frac{\partial T'}{\partial t} \bigg|_i = -u_i \frac{T_{i+1} - T_i}{\Delta x} - \left( \gamma - 1 \right) T_i \left( \frac{u_{i+1} - u_i}{\Delta x} + \frac{u_i}{\rho_i} \frac{\rho_{i+1} - \rho_i}{\Delta x} \right)
\] 

(8)

The right side of the above equation are all known quantities. Next, by taking the first two terms of the Taylor series to find the estimated value \((\rho')_i^{+\Delta t}, (u')_i^{+\Delta t}, (T')_i^{+\Delta t}\), the following equations can be obtained:

\[
\begin{align*}
(\rho')_i^{+\Delta t} &= \rho_i^{+\Delta t} + \frac{\partial \rho'}{\partial t} \bigg|_i \Delta t, \\
(u')_i^{+\Delta t} &= u_i^{+\Delta t} + \frac{\partial u'}{\partial t} \bigg|_i \Delta t \\
(T')_i^{+\Delta t} &= T_i^{+\Delta t} + \frac{\partial T'}{\partial t} \bigg|_i \Delta t
\end{align*}
\] 

(9)

**Correction step:** With the backward difference method to calculate the spatial derivative on the right side of the continuity equation and the estimated values of density, velocity, and temperature, we can calculate the estimated value of the time derivative at time \( t + \Delta t \) and expressed as :
The time-averaged derivative of the above equations are:

\[
\frac{\partial \rho'_{l+t+\Delta t}}{\partial t}_{l} = -\left(\bar{u'}_{l+t+\Delta t} \frac{\rho'_{l+t+\Delta t} - \rho'_{l}}{\Delta x'}\right)_{l-1} + \left(\bar{u'}_{l+t+\Delta t} \frac{\rho'_{l+t+\Delta t} - \rho'_{l}}{\Delta x'}\right)_{l-1} - \left(\bar{\rho'}_{l+t+\Delta t} \frac{\rho'_{l+t+\Delta t} - \rho'_{l}}{\Delta x'}\right)_{l-1}
\]

\[
\frac{\partial u'_{l+t+\Delta t}}{\partial t}_{l} = -\left(\bar{u'}_{l+t+\Delta t} \frac{\bar{u'}_{l+t+\Delta t} - \bar{u'}_{l}}{\Delta x'}\right)_{l-1} - \frac{1}{\gamma} \left(\bar{T'}_{l+t+\Delta t} \frac{\bar{T'}_{l+t+\Delta t} - \bar{T'}_{l}}{\Delta x'}\right)_{l-1} - \left(\bar{u'}_{l+t+\Delta t} \frac{\bar{u'}_{l+t+\Delta t} - \bar{u'}_{l}}{\Delta x'}\right)_{l-1}
\]

\[
\frac{\partial T'_{l+t+\Delta t}}{\partial t}_{l} = -\left(\bar{u'}_{l+t+\Delta t} \frac{T'_{l+t+\Delta t} - \bar{T'}_{l}}{\Delta x'}\right)_{l-1} - (\gamma - 1) \left(\bar{T'}_{l+t+\Delta t} \frac{\bar{T'}_{l+t+\Delta t} - \bar{T'}_{l}}{\Delta x'}\right)_{l-1} - \left(\bar{u'}_{l+t+\Delta t} \frac{T'_{l+t+\Delta t} - \bar{T'}_{l}}{\Delta x'}\right)_{l-1}
\]

The time-averaged derivative of the above equations are:

\[
\frac{\partial \rho'_{l}}{\partial t}_{l} = \frac{1}{2} \left(\frac{\partial \rho'_{l}}{\partial t}_{l} + \frac{\partial \rho'_{l}}{\partial t}_{l} \right)_{t} + \frac{\partial \rho'_{l}}{\partial t}_{l} \Delta t
\]

\[
\frac{\partial u'_{l}}{\partial t}_{l} = \frac{1}{2} \left(\frac{\partial u'_{l}}{\partial t}_{l} + \frac{\partial u'_{l}}{\partial t}_{l} \right)_{t} + \frac{\partial u'_{l}}{\partial t}_{l} \Delta t
\]

\[
\frac{\partial T'_{l}}{\partial t}_{l} = \frac{1}{2} \left(\frac{\partial T'_{l}}{\partial t}_{l} + \frac{\partial T'_{l}}{\partial t}_{l} \right)_{t} + \frac{\partial T'_{l}}{\partial t}_{l} \Delta t
\]

Finally, the corrected values at \(t + \Delta t\) are:

\[
\rho'^{t+\Delta t}_{l} = \rho'^{t}_{l} + \frac{\partial \rho'_{l}}{\partial t}_{l} \Delta t
\]

\[
u'^{t+\Delta t}_{l} = \nu'^{t}_{l} + \frac{\partial u'_{l}}{\partial t}_{l} \Delta t
\]

\[
T'^{t+\Delta t}_{l} = T'^{t}_{l} + \frac{\partial T'_{l}}{\partial t}_{l} \Delta t
\]

3. Results and discussion

3.1. The results in different time steps on the nozzle flow

Figure 1 presents the nozzle shape defined by Eq. (6). Four results in different time steps are presented to show the convergence process of the solution. Red point denotes the position of nozzle throat. The throat lies in the middle of the nozzle. It can be noted that the solution takes about 800 time steps to reach convergence status as the results at time step 800 (denoted as 800dt in the figures) almost coincident with that of the time step 3000. From the results at time step 3000 (denoted as red curves and labelled as 3000dt in the legends), the curves for the flow parameters increase or decrease smoothly, which validate the correctness of the implemented code to some extent.
3.2. The influence of the position of the nozzle throat on the nozzle flow

Four additional nozzles with the throat located in different positions are adopted to find out the influence of throat position on the nozzle flow. The nozzle shapes of the additional four nozzles are defined in Eq. (13). Fig. 3 presents the shapes of the above four nozzles. The red point denotes the position of the nozzle throat.

\[
A(x) = 1 + 2.2(x - 1.0)^2 \\
A(x) = 1 + 2.2(x - 1.3)^2 \\
A(x) = 1 + 2.2(x - 1.8)^2 \\
A(x) = 1 + 2.2(x - 2.0)^2
\]  

(a) Nozzle with \( A(x) = 1 + 2.2(x - 1.0)^2 \) 
(b) Nozzle with \( A(x) = 1 + 2.2(x - 1.3)^2 \)
6

Figure 3. Diagram of four nozzle shapes

(c) Nozzle with \( A(x) = 1 + 2.2(x - 1.8)^2 \)  
(d) Nozzle with \( A(x) = 1 + 2.2(x - 2.0)^2 \)

Figure 4 shows the flow quantities inside five nozzles with different positions of the throat. The values in the nozzle throat are similar. For example, the Mach numbers in the throat are about 1. It is because that the flow will reach the critical status at the minimal cross section. Generally, the position of the nozzle throat influences the velocities at the nozzle exit. The faster the flow reaches the throat, the higher the velocity or Mach number at the exit boundary.

Figure 4. Flow quantities inside five nozzles with different position of throat

3.3. **The flow characteristics inside the convergent nozzle**

A convergent nozzle is employed to find out the characteristics of two nozzles. Fig. 5 presents the nozzle shapes in this part of study. The throat lies in the middle of the throat. The shape curve of this convergent nozzle is identical to that shown in Figure 1. Figure 5 present the flow quantities inside the convergent nozzle. The pressure and density share the identical variation pattern from the inlet to the outlet, which
is the quantity decrease firstly until reaching the throat and then increases. While the velocity increases firstly and reaches the maximal value at the throat. After the throat, the velocity decreases. The Mach number shown in Figure 5(d) shows that the flow is subsonic at the inlet boundary and keeps subsonic inside the whole nozzle.

![Figure 5. Diagram of nozzle shape](image)

![Figure 6. Flow quantities inside a convergent nozzle](image)

4. Conclusions and future work

In this paper, the finite difference method was used to solve the Euler equation to obtain the flow inside the Laval nozzle. An in-house python code was developed to implement the method. The relationship among the velocity, pressure and density in the nozzle flow was found. It is noted that the velocity rose quickly along with the nozzle and reached the top before a rapid decrease. Pressure remained constant initially, which eventually began to drop. Density dropped steadily and had a turning point. With the developed method, the influence of the nozzle throat position is studied. It is observed that the position of the nozzle throat influences the velocities at the nozzle exit. The faster the flow reaches the throat,
the higher the velocity or Mach number at the exit boundary. Overall, this paper researches on accelerating subsonic fluid to supersonic speed. It is commonly used in the aerospace industry, such as airplanes and rockets. Based on the current work, future research can be devoted to investigating the influence of the nozzle throat width on the nozzle flow.

References
[1]. Simscale, (2021) What is CFD, https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/what-is-cfd-computational-fluid-dynamics/
[2]. NAROM, (2018) Nozzle, https://www.narom.no/undervisningsressurser/sarepta/rocket-theory/rocket-engines/nozzle/#:~:text=The%20nozzle%20is%20an%20important,high%20velocity%20but%20low%20pressure.&text=The%20nozzle%20helps%20align%20the,molecules%20to%20the%20same%20direction.
[3]. Balz R.; Nagy I. G.; Weisser G.; Sedarsky D., (2021) Experimental and numerical investigation of cavitation in marine Diesel injectors, International Journal of Heat and Mass Transfer, 169(120933): 1-12
[4]. Guo G., He Z., Wang Q., Lai M., Zhong W., Guan W., Wang J., (2021) Numerical investigation of transient hole-to-hole variation in cavitation regimes inside a multi-hole diesel nozzle, Fuel, 287: 119457
[5]. Sykes D., Turner J., Stetsyuk V., de Sercey G., Gold M., Pearson R., Crua C.I, (2021) Quantitative characterizations of spray deposited liquid films and post-injection discharge on diesel injectors, Fuel, 289(1):119833
[6]. Alya S., Singh R., (2021) Discrete Phase Modeling of the Powder Flow Dynamics and the Catchment Efficiency in Laser Directed Energy Deposition With Inclined Coaxial Nozzles, Journal of Manufacture Science and Engineering, 143(8): 081004
[7]. He C., Liu Y., Gan L., (2021) Dynamics of the jet flow issued from a lobed Nozzle: Tomographic particle image velocimetry measurements, International Journal of Heat and Fluid Flow 89:108795
[8]. Shi G., Nishizawa S., Matsushita T., Kato Y., Kozumi T., Matsui Y., Shirasaki N., Computational fluid dynamics–based modeling and optimization of flow rate and radiant exitance for 1,4-dioxane degradation in a vacuum ultraviolet photoreactor, Water Research, 197: 117086