Investigation of the Influence of Laval Nozzle Throat Width via Euler-based Flow Simulation

Yu’ang Lu*

Department of Mechanical Engineering, Nanjing Tech University, Nanjing, 211816, China

*Corresponding author’s e-mail: 201821138017@njtech.edu.cn

Abstract. The nozzle is a kind of widely used fluid equipment. It is very important to study its internal flow for improving nozzle performance or improving nozzle design. In this paper, the numerical simulation method based on the Euler equation is used to solve the flow in the nozzle. Euler equations are solved by an explicit finite difference method, including discretization, pre-estimation and correction steps. Boundary treatment for the subsonic inlet and supersonic outlet are discussed. The flow characteristics inside a Laval nozzle are analysed based on the numerical results. The influence of Laval nozzle throat width on the nozzle flow is investigated by comparing three nozzles. It is observed that the dimensionless pressure, temperature, and density of the three nozzles have similar trends. While the maximum Mach number at the outlet decreases with the increase of the throat width. In short, the throat width should be decreased if a larger outlet speed of the flow is needed.

1. Introduction

Air and water are common fluids. Fluid mechanics is a subject of studying fluids, mainly focusing on the static and moving state of the fluid, and the interaction and flow law when there is relative motion between fluid and solid boundary. It is mainly based on Newton’s law of motion and the law of conservation of mass, often using the knowledge of thermodynamics, and sometimes using the basic laws of macro electrodynamics, constitutive equations and the basic knowledge of physics and chemistry [1]. The nozzle is a kind of common equipment to control the velocity of the fluid, and it is widely used in rockets and engines.

Because of its wide application, many scholars have carried out relevant research on it. In the numerical calculation of a low thrust measurement system for Laval nozzle, by Yonghua Lu [2], the flow state in a pipe with different stagnation pressure is analyzed theoretically and the applicability of the model is verified. According to the position of baffle and inlet total pressure position, the Laval nozzle and its extended flow field are simulated, and the influence of baffle on the nozzle and its extended flow field is analyzed. In the theoretical analysis and numerical simulation of fluid flow in the nozzle , by Tao Liu [3], the relationship between the cross-sectional area of the pipe in the nozzle and several state parameters is analyzed and the numerical simulation of the characteristic value of gas flow field in a pipeline is feasible in practice. The comparison between simulation and theoretical analysis shows that the simulation is correct, and the numerical simulation of gas flow field characteristics in the pipeline is feasible in practical work. In calculating and analyzing flow state parameters in Laval nozzle, by Chaohua Yuan [4], the flow parameters of the axial distribution in Laval nozzle at supersonic input and supersonic output are analyzed. The results show that the Mach
number in the throat reaches the minimum, and the Mach number in the throat increases with the increase of the inlet Mach number.

In this paper, the Euler-based numerical simulation method analyzes the flow inside the Laval nozzle. With the developed method, the influence of the nozzle throat width on the nozzle flow is investigated schematically. The remainder of this paper is organized as follows. In Section 2, the adopted method for nozzle flow simulation is introduced in detail. In Section 3, the simulation results are presented and discussed. Finally, conclusions are drawn in Section 4.

2. Method

In this method to obtain the flow inside a nozzle is detailed. Euler equation is one of the main equations used to study the flow in the Laval nozzle. I mainly studied the one-dimensional Euler equation, including constant differential calculation of the one-dimension Euler equation. Firstly, how to solve the one-dimensional Euler equation is presented in the following content.

Euler equation in 1-D is presented in the followings [5]:

$$
\frac{\partial (\rho A)}{\partial t} + \rho A \frac{\partial V}{\partial x} + \rho V A \frac{\partial A}{\partial x} + VA \frac{\partial \rho}{\partial x} = 0
$$

$$
\rho \frac{\partial V}{\partial t} + \rho V \frac{\partial V}{\partial x} = -R \left( \frac{\partial T}{\partial x} + T \frac{\partial \rho}{\partial x} \right)
$$

$$
\rho c_v \frac{\partial T}{\partial t} + \rho V c_v \frac{\partial T}{\partial x} = \rho RT \left[ \frac{\partial V}{\partial x} + V \frac{\partial (\ln A)}{\partial x} \right]
$$

where \( \rho, A, V, T, R, c_v \) represent density, nozzle cross-sectional area, velocity, temperature, gas constant, and constant volume specific heat capacity, respectively. The first line in the Eq. (1) represents the continuity equation. The second line is the momentum equation in 1-D form. The energy equation is presented in the third line.

In this paper, the finite difference expression of MacCormack’s explicit method is used because the analytical solution of the above formula cannot be obtained. To carry out the finite difference calculation, the nozzle is divided into many discrete grid points along the x-axis. These grid points are evenly distributed along the x-axis, and the spacing is \( \Delta x \), as shown in Figure 1. In Figure 1, a total of \( N \) grid points are distributed on the x-axis. The first grid point which is labeled as 1 is the grid point in the inlet, and the last node which is labeled as \( n \) is located at the nozzle exit. If \( i \) is used to representing any grid point, then \( i - 1 \) and \( i + 1 \) are two adjacent grid points. The time marching method uses the known flow field variables at time \( t \) to solve the flow field variables at time \( t + \Delta t \) through the differential equation. McCormack’s method is a predictor-corrector for time marching.

Figure 1 Distribution of grid points along the nozzle

The first step is the pre-estimation procedure. In MacCormack’s method, the forward differential scheme is employed to calculate the spatial derivatives. In detail, it is expressed as:
\[ \left( \frac{\partial \rho}{\partial t} \right) \bigg|_j = -V'_i \left( \rho'_{i+1} - \rho'_i \right) - \rho'_i \left( V'_{i+1} - V'_i \right) - \rho'_i \frac{\ln A_{i+1} - \ln A_i}{\Delta x} \]
\[ \left( \frac{\partial V}{\partial t} \right) \bigg|_j = V'_i \left( \frac{V'_{i+1} - V'_i}{\Delta x} \right) - \frac{1}{\gamma} \left( \frac{T'_{i+1} - T'_i}{\Delta x} + \frac{T'_i}{\rho'_i} (\rho'_{i+1} - \rho'_i) \right) \]
\[ \left( \frac{\partial T}{\partial t} \right) \bigg|_j = -V'_i \left( \frac{T'_{i+1} - T'_i}{\Delta x} \right) - (\gamma - 1)T'_i \left( \frac{V'_{i+1} - V'_i}{\Delta x} + V'_i \frac{\ln A_{i+1} - \ln A_i}{\Delta x} \right) \]

Then the pre-estimated value of the \( \rho, V \) and \( T \) is defined as:
\[ \rho^{i+1}_{\Delta t} = \rho'_i + \left( \frac{\partial \rho}{\partial t} \right) \bigg|_j \Delta t \]
\[ V^{i+1}_{\Delta t} = V'_i + \left( \frac{\partial V}{\partial t} \right) \bigg|_j \Delta t \]
\[ T^{i+1}_{\Delta t} = T'_i + \left( \frac{\partial T}{\partial t} \right) \bigg|_j \Delta t \]

where \( \rho'_i, V'_i \) and \( T'_i \) are known quantities at time \( t \). The time derivative in the equation is directly given by Eq. (2). The corrections step is introduced next. In this case, the spatial derivatives are calculated by the estimated backward difference, which are:
\[ \left( \frac{\partial \rho}{\partial t} \right) \bigg|_{av} = -F^{i+1}_{i+\Delta} \frac{\rho_{i+\Delta} - \rho_{i}}{\Delta x} - \rho_{i} \frac{\ln A_{i+1} - \ln A_i}{\Delta x} \]
\[ \left( \frac{\partial V}{\partial t} \right) \bigg|_{av} = -F^{i+1}_{i+\Delta} \frac{V_{i+\Delta} - V_i}{\Delta x} - \frac{1}{\gamma} \left( \frac{T_{i+\Delta} - T_i}{\Delta x} + \frac{T_i}{\rho_{i+\Delta}} \rho_{i+\Delta} - \rho_{i} \right) \]
\[ \left( \frac{\partial T}{\partial t} \right) \bigg|_{av} = -F^{i+1}_{i+\Delta} \frac{T_{i+\Delta} - T_i}{\Delta x} - (\gamma - 1)T_i \left( \frac{V_{i+\Delta} - V_i}{\Delta x} + V_i \frac{\ln A_{i+1} - \ln A_i}{\Delta x} \right) \]

Then the average value of the time-related derivatives is given as:
\[ \left( \frac{\partial \rho}{\partial t} \right) \bigg|_{av} = \frac{1}{2} \left[ \left( \frac{\partial \rho}{\partial t} \right) \bigg|_j + \left( \frac{\partial \rho}{\partial t} \right) \bigg|_{i+\Delta} \right] \]
\[ \left( \frac{\partial V}{\partial t} \right) \bigg|_{av} = \frac{1}{2} \left[ \left( \frac{\partial V}{\partial t} \right) \bigg|_j + \left( \frac{\partial V}{\partial t} \right) \bigg|_{i+\Delta} \right] \]
\[ \left( \frac{\partial T}{\partial t} \right) \bigg|_{av} = \frac{1}{2} \left[ \left( \frac{\partial T}{\partial t} \right) \bigg|_j + \left( \frac{\partial T}{\partial t} \right) \bigg|_{i+\Delta} \right] \]

Finally, the corrected value at the time step \( t + \Delta t \) is obtained as:
\[ \rho^{i+1}_{\Delta t} = \rho'_i + \left( \frac{\partial \rho}{\partial t} \right) \bigg|_{av} \Delta t \]
\[ V^{i+1}_{\Delta t} = V'_i + \left( \frac{\partial V}{\partial t} \right) \bigg|_{av} \Delta t \]
\[ T^{i+1}_{\Delta t} = T'_i + \left( \frac{\partial T}{\partial t} \right) \bigg|_{av} \Delta t \]

Another very important aspect of the numerical method is the boundary condition. Without accurate physical description and proper numerical treatment of the boundary conditions, it is impossible to obtain the correct numerical solution of the flow problem.

Subsonic inflow boundary (grid point 1). On this boundary, a variable must be allowed to change. We choose to change the velocity \( V_1 \). From the physical point of view, the mass flow through the
nozzle should be able to adjust to a suitable steady state. As part of this adjustment, it is reasonable to allow $V_3$ to vary. When the value of $V_3$ changes with time, we need to use the information provided by the flow field solution on the interior point (the interior point is the point that is not on the boundary). Here, using the values of grid points 2 and 3, we use the linear extrapolation method to calculate $V_1$, as shown in Figure 2. The slope of the linear extrapolated line is determined by grid point 2 and grid point 3.

The slope of a straight line $k$ is shown as follows:

$$ k = \frac{V_3 - V_2}{\Delta x} $$  \hspace{1cm} (7)

Extrapolate the value of $V_1$ according to the slope, that is:

$$ V_1 = V_2 - \frac{V_3 - V_2}{\Delta x} \Delta x $$  \hspace{1cm} (8)

Or

$$ V_1 = 2V_2 - V_3 $$  \hspace{1cm} (9)

Except $V_1$, other flow field parameters are given. Since grid point 1 is considered to be located in the parking room, it can be specified that the density and temperature at point 1 are stagnation parameters, $\rho_0$ and $T_0$. These two values are fixed and do not change with time. In terms of dimensionless quantity:

$$ \begin{align*}
\rho_1 &= 1 \\
T_1 &= 1
\end{align*} $$  \hspace{1cm} (10)

Supersonic exit boundary (grid point N): Here, all flow field parameters must be allowed to change. Therefore, the flow field parameters at the interior point are still needed to be used for linear extrapolation, expressed by dimensionless variables, which is:

$$ \begin{align*}
V_N &= 2V_{N-1} - V_{N-2} \\
\rho_N &= 2\rho_{N-1} - \rho_{N-2} \\
T_N &= 2T_{N-1} - T_{N-2}
\end{align*} $$  \hspace{1cm} (11)

Nozzle shape $A = A(x)$ is given and remains unchanged. An area distribution determined by quadratic function is given here:

$$ A = 1 + 2.2(x - 1.5)^2 $$  \hspace{1cm} (12)

Note that $x = 1.5$ represents the throat of the nozzle. When $x < 1.5$, it is the convergent segment, and when $x > 1.5$, it is the divergent segment. In order to start time advance, the initial conditions of $\rho$, $T$ and $V$ must be given. That is to say, the values of $\rho$, $T$ and $V$ must be given when $t = 0$, which are functions of $x$. When $t = 0$, the equation can be given:
\[
\begin{align*}
\rho &= 1 - 0.314x \\
T &= 1 - 0.2314x \\
V &= (0.1 + 1.09x)\sqrt{T} 
\end{align*}
\] (13)

3 Results and discussion

3.1 Illustration of the Euler-based flow simulation

In the numerical simulation of the nozzle with shape function being \( A = 1 + 2.2(x - 1.5)^2 \), 31 nodes along the x-axis are employed and the maximum time step is specified as 2000 to achieve steady solutions. Figure 3 shows the dimensionless mass flow rate distribution at three steps, the first, 500th, 1000th step. It can be noted that the mass flow rate along the nozzle takes diverse values at the first-time step (the blue line in Figure 3). With the increase of the time step, the mass flow rate distribution becomes flat (the red line in Figure 3), indicating the predicted flow inside the nozzle reaches the steady state. At the 1000th step, the mass flow rate almost takes the identical value (the green line in Figure 3), which means the iterative solution reaches convergence. This can also be confirmed by the convergence histories of the dimensionless density, temperature, pressure, and Mach number shown in Figure 4. All those quantities reached a steady state which can be noted from the flat curves in Figure 4.

Figure 3 Dimensionless mass flow rate distribution along the nozzle at three steps

Figure 4 Convergence histories of the dimensionless density, temperature, pressure, and Mach number
The steady state of the dimensionless density, temperature, pressure, and Mach number inside the nozzle are presented in Figure 5. The dimensionless pressure decreases along with the nozzle. Specifically, it decreases slowly near the inlet, then the slope of the pressure curve increases fast, and takes the maximum value at the throat position. The trends of the temperature and density curve are identical to that of the pressure curve. While the Mach number increases along the nozzle and reaches its peak value at the outlet. This means that the increase of the velocity results from the decrease of the pressure, or the kinematic energy is transformed from the enthalpy. The transformation results from the variation of the cross-sectional area along with the nozzle.

![Figure 5 Steady state of the dimensionless pressure, temperature, density, and Mach number inside the nozzle](image)

3.2 Influence of nozzle throat on the nozzle flow

Two additional nozzles are used to investigate the influence of nozzle throat width on the nozzle flow and their flow is predicted with the Euler-based flow simulation method. For convenience, the three nozzles are denoted as Nozzle1, Nozzle2, and Nozzle3 respectively with the increase of the nozzle throat width. The nozzle used in the last subsection is Nozzle2. The grid nodes and the number of time steps are identical to that used in the last subsection. The shape functions of the additional two nozzles, i.e. Nozzle1 and Nozzle3, are depicted in the Eq. (14) and their shapes are presented in Figure 6(a) and (b).

\[ \text{Nozzle1: } A = 0.5 + 2.2(x - 1.5)^2 \]
\[ \text{Nozzle3: } A = 1.5 + 2.2(x - 1.5)^2 \]  

(14)
Nozzle1 shape with $A = 0.5 + 2.2(x - 1.5)^2$  
Nozzle3 shape with $A = 1.5 + 2.2(x - 1.5)^2$

Figure 6 Shapes of additional nozzles

The steady state of the dimensionless pressure, temperature, density, and Mach number inside the Nozzle1 and Nozzle3 are presented in Figure 7 and Figure 8, respectively. Overall, those dimensionless quantities (including pressure, temperature, and density) of Nozzle1 and Nozzle3 have similar distributions along with the nozzle compared with that of Nozzle2. The Mach number distributions have similar trends but with different variation ranges. For convenience, Figure 9 presents the Mach number distributions only. With the increase of the throat width, the maximum Mach number (i.e. the Mach number at the outlet) decreases from 4 for Nozzle1 to 3 for Nozzle3. With this observation, if a larger outlet speed of the flow is needed, the throat width should be decreased, vice versa.

Figure 7 Steady state of the dimensionless pressure, temperature, density, and Mach number inside the Nozzle1
Figure 8 Steady state of the dimensionless pressure, temperature, density, and Mach number inside the Nozzle3

Figure 9 Steady Mach number distribution inside the nozzles

(a) Nozzle1 (b) Nozzle2 (c) Nozzle3

4 Conclusion and future work
In this paper, the numerical simulation method based on the Euler equation is used to solve the flow in the Laval nozzle. MacCormark’s explicit finite difference method is employed to solve the Euler equations, including the pre-estimation and correction two steps generally. The treatment of boundary conditions for the subsonic inlet and supersonic outlet are detailed. The solving process is first discussed to verify the effectiveness of the implemented code. The flow characteristics inside the Laval nozzle are analysed from the numerical results. Then flow quantities inside three nozzles with different throat widths are compared. It is found that the dimensionless quantities (including pressure, temperature, and density) of the three nozzles have similar trends. The Mach number distributions they have similar trends but with different variation ranges. With the increase of throat width, the maximum Mach number at the outlet decreases. In other words, if a larger outlet speed of the flow is needed, the throat width should be decreased.

References
[1] Philip M. G. (2016) Fundamentals of Fluid Mechanics. Wiley, New York.
[2] Hu X., Gao W., Liu T., Lin W. (2015) Theoretical analysis and numerical simulation of fluid flow in nozzle. Journal of Yunan Normal University, Vol.35 No.1:11-17
[3] Lu Y., Wei W., Shao F., Sun Y. (2016) Numerical calculation of low thrust measurement system for Laval nozzle. Journal of Mechanical&Electrical Engineering, Vol.33 No.10:1180-1192
[4] Liu K., Zhu M., Gao Y., Lv G., Wang Z. (2007) Numerical simulation of flow character in Laval nozzle. Energy for metallurgical industry, Vol.26 No.4: 20-23

[5] Anderson J. D. (1995) Fundamentals and applications of computational fluid dynamics. University of Maryland, Maryland

[6] Liu Z., Zhang T. (2009) Calculation and analysis of gas flow in Laval micro nozzle. Journal of Aerospace Power, Vol.24 No.7: 1556-1563

[7] Yao Y., Zhang J. (2006) Study on flow field of high speed mixture in Laval nozzle. Journal of Computer Simulation, Vol.23 No.1: 195-199

[8] Xiao Y., Xu H. (2010) Numerical study of steam condensation flow in Laval nozzle. Turbine Technology, Vol.52 No.4: 284-287

[9] Cui Q., Yang W., Wang Z., Zhou J. (2012) Experimental study and numerical simulation of gas flow in micro nozzle, Vol.12 No.4: 12-15, 35

[10] Chen W., Wang Q., Yin L. (2005) Comparative study of particle simulation and Navier Stokes numerical simulation of micro nozzle flow. Chinese Journal of Computational Mechanics, Vol.22 No. 6: 677-680

[11] Ding Y., Yao C., He F. (2004) Study on gas flow characteristics in micro nozzle. Engineering Mechanics, Vol.22 No.3: 190-195