Numerical Simulation of Flow Field in the Laval Nozzle Based on Euler Equation

Zewei Zhang\textsuperscript{1, a, †}, Baozhao Yi\textsuperscript{2, b, †}, Zhexuan Tan\textsuperscript{3, c, †}

\textsuperscript{1}Department of Applied Mechanics and Aerospace Engineering, Waseda University, Tokyo, 169-8555, Japan
\textsuperscript{2}School of Mechanical, Electrical & Information Engineering, Shandong University, Weihai, 264209, China
\textsuperscript{3}School of Civil Engineering, Central South University, Changsha, 410083, China
\textsuperscript{a}ctki49@akane.waseda.jp, \textsuperscript{b}201800800319@mail.sdu.edu.com, \textsuperscript{c}csutzxwd@csu.edu.com.
\textsuperscript{†}These authors contributed equally.

Abstract. A nozzle is a versatile fluid device that is utilized to control the characteristics of a fluid flow. In order to enhance the performance or better the design of the nozzle, it is critical to carry out extensive research into its inner flow. Thus, the investigation of the nozzle flow remains an important challenge. This article used a numerical simulation method based on Euler equations to solve the internal flow of the divergent, convergent-divergent, and convergent nozzles. Specifically, we simplify all three equations of the one-dimensional Euler equation (the continuity equation, the momentum equation, and the energy balance equation) by assuming a constant nozzle shape, then discretizing them using a central differential scheme. After conducting systematic research on the influence on the internal nozzle flow due to the change of nozzle shape, we found that the mass flow rate varied as it fluctuated and peaked at the middle point of the Laval nozzle. However, with the nozzle becomes flatter, the mass flow rate appeared to restore conserved. Moreover, it appeared that the flatter the Laval nozzle is, the smaller acceleration and smaller decrease of the pressure of the fluid flow gets. In short, the results suggested that it is dependable to apply this numerical method when analysing a relatively flat Laval nozzle.

1. Introduction
Computational fluid dynamics (CFD) is an important part of fluid mechanics and plays an important role in various fields of fluid mechanics. CFD originated in aircraft manufacturing, but now it has applied in ranges of aerospace, transportation, shipbuilding, meteorology, petrochemical, and other fields. With the rapid development of computer technology, the CFD will also get a bigger breakthrough. Then it will greatly improve the efficiency of problem-solving. A Laval nozzle is a nozzle that accelerates the fluid from the subsonic stage to the supersonic stage. In the convergent section (subsonic phase), the velocity of fluid increases constantly. The fluid velocity increases in the divergent section (supersonic stage). Laval nozzles are widely used in supersonic aircraft [1,2] and rocket engines [3].

Wang et al. [4] used the CFD method to simulate the seven work conditions of Laval nozzle flow. Combined with one-dimensional isentropic flow theory, this team analysed the inside condition of the nozzle flow. Besides, the finite volume method was adopted to solve the Navier-Stokes’s equations. As
for flow field initialization of flow calculation, the author combined one-dimensional isentropic, full multigrid initialization, and extrapolation to solve it, which greatly improved the simulation efficiency. Zhang et al. [5] used Solidworks2016 to carry out the three-dimensional modelling of the Laval nozzle, and then the ANSYS19.0 Workbench-CFX was employed to analyse the flow condition of the Laval nozzle to obtain pressure, velocity, and the Mach number. In conclusion, for subsonic flow, the pressure decreased while the velocity and Mach number increased with the decline of cross-sectional area, vice versa. As for supersonic flow, it could be witnessed that the pressure grew, whereas the velocity and Mach number declined with the reduction of cross-sectional area. Yuan et al. [6] used the ANSYS CFX to simulate the flow distribution in different shapes of Laval nozzle, and they got the natural gas flow distribution line chart in different nozzles. The studies of the different lines concluded that the inlet diameter had a small influence on the flow distribution in the nozzle. In contrast, the throat and outlet diameter were the main reasons for influencing the flow distribution. Among them, small throat diameter had an obvious impact on the entire flow distribution. However, large throat diameter significantly only affected the flow distribution in the divergent section. Besides, the flow distribution of the throat and divergent section was largely determined by the outlet diameter. Wang et al. [7] implemented the pressure boundary for the direct simulation Monte Carlo method to analyse the gas flows in two different types of micro Laval nozzles. Subsequently, the flow characteristics under different outlet pressure boundaries with the same inlet pressure were compared. On top of that, the influence of the outlet boundary on the flow field was investigated under a vacuum outlet and a given low pressure outlet boundary condition. Wang et al. [8] established the numerical model of the Laval nozzle and employed the analytical method, The CFX simulation, and numerical method, respectively, to analyse the flow in convergent section and in divergent section. Finally, the research showed that the results by these three methods were in common. Gao et al. [9] firstly determined the shape of the Laval nozzle based on thermodynamic calculation. Subsequently, they adopted Fluent to simulate the flow field in the nozzle. On top of that, they changed the shapes of the nozzle and analysed the influence of different geometry of divergent and convergent sections on exit velocity. Finally, it could be found in this research that the exit diameter had a huge influence on the exit velocity. In the end, they found the best exit diameter from a range of experimental data and improved the geometrical size of the nozzle.

This research mainly focused on simulating the Laval nozzle numerically based on both the Euler equation and comparing different nozzle shapes. For the Euler equations, we firstly used the finite differences method to simplify it, and then introduced the inflow and outflow boundary to draw the diagrams of mass flow rate pressure and Mach number of the flow field in the Laval nozzle.

2. Method

2.1. 1D Euler’s Equation

1D Euler’s Equation in the conservative form is given by

$$\frac{\partial u}{\partial t} + \frac{\partial F}{\partial x} = 0$$

(1)

where $U = \begin{bmatrix} \rho \\ \rho u \\ E \end{bmatrix}$, $F = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ u(E + p) \end{bmatrix}$

(2)

where $\rho, u, p, E$ denotes density, flow velocity, pressure, and total energy per unit volume$(\frac{1}{2} \rho u^2 + \rho e)$, respectively. $e$ means the specific internal energy here. The 1D Euler’s equations include the continuity equation, the momentum equation, and the energy balance equation in fluid dynamics. Meanwhile, 1D Euler’s equation can be regarded as the transformed Navier-Stokes equations, which have ignored the influence of the viscosity term. For nozzle flow, the continuity equation, the momentum equation, and the energy balance equation are still satisfied. Considering the cross section change $dS$ along with the nozzle, the continuity equation can be written as:

$$\frac{\partial (\rho S)}{\partial t} + \frac{\partial (\rho S u)}{\partial x} = 0$$

(3)
Then, by assuming that the shape of the nozzle is constant, Eq. (3) can be transformed into the conservative form, which is shown as:

$$\frac{\partial p}{\partial t} + S \frac{\partial (\rho u)}{\partial x} + \rho u \frac{\partial S}{\partial x} = 0$$  \hspace{1cm} (4)

It holds the same for the momentum equation and energy balance equation.

$$\frac{\partial (\rho u)}{\partial t} + \frac{\partial (\rho u^2 + p)}{\partial x} + \rho u \frac{\partial S}{\partial x} = -\frac{\partial p}{\partial x}$$  \hspace{1cm} (5)

$$\frac{\partial E}{\partial t} + S \frac{\partial (u(E + p))}{\partial x} + u(E + p) \frac{\partial S}{\partial x} = 0$$  \hspace{1cm} (6)

After the transformation, Eq. (5) and Eq. (6) become:

$$\frac{\partial (\rho u)}{\partial t} + S \frac{\partial (\rho u^2 + p)}{\partial x} + \rho u^2 \frac{\partial S}{\partial x} = 0$$  \hspace{1cm} (7)

$$\frac{\partial E}{\partial t} + S \frac{\partial (u(E + p))}{\partial x} + u(E + p) \frac{\partial S}{\partial x} = 0$$  \hspace{1cm} (8)

By summarizing these equations, we can obtain:

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{S}{S'} G = 0$$  \hspace{1cm} (9)

$$G = \begin{bmatrix} \rho u \\ \rho u^2 \\ u(E + p) \end{bmatrix}$$  \hspace{1cm} (10)

where \(S\) denotes the function describing the cross section of the nozzle and \(S'\) denotes the derivative of \(S\). Eq. (9) is the 1D Euler’s equation for nozzle flows in the conservative form. This research uses a central differential scheme to discretize the formula above. The central differential scheme is written as:

$$\frac{\partial U}{\partial t} = \frac{U_{i+1} - U_{i-1}}{2\Delta t}, \quad \frac{\partial F}{\partial x} = \frac{F_{i+1} - F_{i-1}}{2\Delta x}$$  \hspace{1cm} (11)

By putting Eq. (5) into Eq. (3) gives:

$$\frac{u^{j+1} - 0.5s(u_{i+1}^{j} + u_{i-1}^{j})}{\Delta t} + \frac{F_{i}^{j} - F_{i-1}^{j}}{2\Delta x} + \frac{S}{2S'} (G_{i+1}^{j} + G_{i-1}^{j}) = 0$$  \hspace{1cm} (12)

$$U_{i}^{j+1} = 0.5(U_{i+1}^{j} + U_{i-1}^{j}) - \frac{\Delta t}{2\Delta x} \left( F_{i+1}^{j} - F_{i-1}^{j} \right) + \frac{\Delta t S}{2S'} \left( G_{i+1}^{j} + G_{i-1}^{j} \right)$$  \hspace{1cm} (13)

where \(i\) means the discretization of distance and \(j\) means the discretization of time. In addition, to avoid the problems that may cause the overflow during the calculation, this research uses matrix \(U\) to express matrix \(F\) and \(G\). The relationships between \(U\) and \(F/G\) are shown below. Note that \(U_1, U_2, U_3\) in the following represents one of the factors in matrix \(U\).

$$U = \begin{bmatrix} \frac{U}{u_1} \\ \frac{U}{u_2} \end{bmatrix}$$  \hspace{1cm} (14)

$$F = \begin{bmatrix} \frac{u_2}{u_1} \frac{u_2}{u_1} \left( U_3 - \frac{U_2}{2u_1} \right) \\ \frac{u_2}{u_1} \left( U_3 + (y - 1) \left( U_3 - \frac{U_2}{2u_1} \right) \right) \end{bmatrix}$$  \hspace{1cm} (15)

$$G = \begin{bmatrix} \frac{u_2}{u_1} \left( U_3 + (y - 1) \left( U_3 - \frac{U_2}{2u_1} \right) \right) \\ \frac{u_2}{u_1} \left( U_3 + (y - 1) \left( U_3 - \frac{U_2}{2u_1} \right) \right) \end{bmatrix}$$  \hspace{1cm} (16)
2.2. Steps of numerical calculation

The equations shown above are numerically solved in the following steps as depicted in Figure 1. Firstly, the shape function of the nozzle is defined. Then the discretization procedure is carried out. In the third step, the initial and boundary conditions are specified. After that, Eq. (9) is utilized to update the matrix $U$. Finally, the converged solutions are outputted. The detailed setting used in this study are discussed in the following.

Figure 1. Five steps for solving the Euler equations

This research defines different shapes of nozzles in step (1). The function of $S$ is defined as:

$$S = 1 + \alpha(x - 1)^2$$

where $\alpha$ denotes the coefficient of nozzles and ranges from 0 to 2. Specifically, four nozzles with $\alpha=0.5$, 1, 1.5, 2 are used in this study. The shapes of the four nozzles are displayed by Figure 2. It can be noted that with the increase of $\alpha$, the cross-section area of the inlet and outlet increases, but the width of the nozzle throat holds unchanged.

Figure 2. the shapes of nozzles when $\alpha=0.5, 1, 1.5, 2$
During Step (2), the discretization of space and time is provided. The discretization cannot be too fine because of the extremely slow speed of getting convergent. It is inappropriate to get a precise solution to give a rough discretization as well. In this study, the range of space and time is \((0, 2)\) and \((0, 20)\), and the grid number is 128. Correspondingly, the step size of space and time is given as 0.015625 and 0.0046875. The initial condition provided in Step (3) becomes:

\[
U_1 = \rho \begin{cases} 
1 & -1 \leq x < -\frac{1}{3} \\
1 - 0.3x & -\frac{1}{3} \leq x \leq 1 
\end{cases}
\] (18)

\[
U_2 = \rho u = \begin{cases} 
0.71 & -1 \leq x < -\frac{1}{3} \\
0.71(1 - 0.3x) & -\frac{1}{3} \leq x \leq 1 
\end{cases}
\] (19)

\[
U_3 = E = \begin{cases} 
2.752 & -1 \leq x < -\frac{1}{3} \\
2.752 - 0.2x & -\frac{1}{3} \leq x \leq 1 
\end{cases}
\] (20)

Meanwhile, as for the boundary condition, one physical boundary condition and two numerical boundary conditions are implemented at the inlet \((x = -1)\), while three numerical boundary conditions are implemented at the outlet \((x = 1)\). The physical boundary condition is a kind of boundary condition that has a fixed value at the boundary. The numerical boundary condition is that the values extrapolate the value at the boundary in the numerical domain. Based on the physical intuition, the change happening to the downstream generally does not affect the upstream. In other words, any boundary conditions at the outlet would not change the solution within the nozzle. Nevertheless, the only effect it brings to this numerical simulation is the smoothness of the solution at the outlet. Therefore, in order to avoid the blow-up of the solution, it is better to use numerical boundary conditions at the outlet.

\[
\begin{align*}
(U_1)_{i=0}^j &= \rho_{in} = 1 \\
(U_2)_{i=0}^j &= 2(U_2)_{i=1}^j - (U_2)_{i=1}^1 \\
(U_3)_{i=0}^j &= \frac{p_{in}}{\gamma - 1} + 0.5\left((U_2)_{i=0}^j\right)^2 \quad \text{at } x = -1 \\
\rho_{in} &= 1 \\
\gamma &= \frac{1.4p_{in}}{\rho_{in}} = 1.183
\end{align*}
\] (21)

\[
\begin{align*}
(U_1)_{i=N-1}^j &= 2(U_1)_{i=N-2}^j - (U_1)_{i=N-3}^j \\
(U_2)_{i=N-1}^j &= 2(U_2)_{i=N-2}^j - (U_2)_{i=N-3}^j \quad \text{at } x = 1 \\
(U_3)_{i=N-1}^j &= 2(U_3)_{i=N-2}^j - (U_3)_{i=N-3}^j
\end{align*}
\] (22)

where \(N\) denotes the number of the grid which equals to 128 in this research; \(c_{in}\) denotes the sound speed at the inlet. Step (4) uses Eq. (7) to upgrade the solution of matrix \(U\) and then outputs the results of \(U\) at \(t = 20\) in step (5).

3. Results and Discussions
Figure 3 shows converging process when \(a=0.5, 1, 1.5, 2\). The flow velocity changes from the initial status to the steady status in this figure. The arrow direction indicates the convergent direction. Every line is drawn in Figure 3 every 100 times steps. According to the number of lines drawn before getting convergent from Figure 3 (a) to Figure 3 (d), we can know that flatter nozzles could have a lower convergent speed.
Figure 3. Convergence histories of velocity in the nozzle at different times with $a=0.5$, $1$, $1.5$, $2$

As shown in Figure 4(a) to (d), the distribution of Mach number along the Laval nozzle is consistent with the physical intuition, which is increasing from subsonic area to supersonic area. However, the mass flow rate does not conserve completely and peaks at the middle point of the Laval nozzle. As for the influence of nozzle shapes on the solution, the mass flow rate would become more conserved when the nozzle shape gets flat. In other words, the peak appearing at the middle point of the nozzle would decline with the decrease of $a$. This phenomenon shows that the flatter nozzle possibly brings higher reliability to the solution from the perspective of mass flow rate conservation. Meanwhile, the Mach number at the outlet goes down and rises at the inlet with the decrease of $a$. It means that a flatter Laval nozzle provides smaller effects of acceleration, which also satisfies the physical intuition. As for the change of pressure, due to the weaker acceleration effect when $a = 0.5$, the decrease of pressure gets smaller.
Figure 4. Mach number, pressure, and mass flow rate when $a = 0.5, 1, 1.5, 2$

Based on the above results, it is possible to judge how reliable the numerical solution could be and to decide about whether to use this numerical method or not. For a relatively bent Laval nozzle, the error of the mass flow rate cannot be ignored, while it is not problematic to apply this method to a relatively flat Laval nozzle to simulate the Mach number distribution along with the nozzle.

4. Conclusions and Future Work
This paper used the Euler-based numerical simulation method to analyse the flow characteristics inside the Laval nozzle. Besides, the influence of nozzle shapes on velocity, Mach number, pressure, and mass flow rate of flow in nozzles is investigated schematically. The first observation is that the smaller the nozzles' coefficient, the lower the convergent velocity is. Moreover, the distribution of the Mach number in the nozzle meets the physical intuition. And the Mach number declines at the outlet, whereas it increases at the inlet. However, the mass flow rate does not absolutely conserve, rising the peak at the middle point of the nozzle. When the nozzle coefficient decreases, the mass flow would be more conversed, which means that the solution would become more dependable. As for the pressure, it falls through the whole process. For future work, we will focus on the influence of the throat width on the nozzle flow.

References
[1] Zhang, L., Guo, X., Zhao, E., He, F. (2017) The brief introduction to the development of computational fluid dynamics (CFD). In: The 23rd Annual Conference of Beijing Society of Theoretical and Applied Mechanics. Beijing. pp. 119–121.
[2] Wang, B. (2018) The research on development and application of computational fluid dynamics. China Southern Agricultural Machinery., 49: 145.
[3] Yu, J., Feng, X. (2016) The research summary of development in computational fluid dynamics. Modern Manufacturing Technology and Equipment., 06: 25–28.
[4] Wang, P., Li, C., Chen, B. (2012) Laval Nozzle Flow Analysis Based on CFD Computation. Aeronautical Computing Technique., 42: 60–62.
[5] Zhang Q. (2019) Simple analysis of the implementation of Laval nozzle based on CFX. Technology and Market., 26: 81–86.
[6] Yuan, P., Xu, W., Lv, Y., Fu, Y. (2015) Numerical simulation of the influence of different geometry of three-dimension nozzle on the flow field distribution. Oil-Gas Field Surface Engineering., 34: 31–33.
[7] Wang, M., Chen, Z., Li, Z. (2003) Simulation and analysis for gas flow and heat transfer in micro nozzle. Micronanoelectronic Technology., 7/8: 61–64.
[8] Zhao, H., Liu, K., Yu, X. (2013) Numerical Simulation of Laval Nozzle. Applied Mechanics and Materials., 397–400: 266–269.
[9] Gao, Q., Tang, H., Wang, Z., He, Y. (2015) Numerical simulation and structure optimization of supersonic nozzle based on Fluent. Manufacturing Automation., 37: 88–108.