Nozzle Flow Simulation by Small Disturbance Approximation and Euler Method

Fanrui Cheng1, †, Yang Xu2, a, *, †, Wenbo Zhou3, b, *, †

1Faculty of Science, Engineering and Technology, Swinburne University of Technology, Hawthorn, Victoria, 3122, Australia
2Department of Energy and Power Engineering, Nanjing University of Science and Technology, Nanjing, 210094, China
3Department of Mechanical and Engineering, Huazhong University of Science and Technology, Wuhan, 430074, China
*a Corresponding author’s e-mail: a xuyang-sia@njust.edu.cn, b wenbozhou@hust.edu.cn.
†These authors contributed equally.

Abstract. The nozzle is a kind of equipment with a wide range of usage in the aerospace industry, of which its performance is vital for rocket engines by changing the geometry of the inner wall of the pipe section to accelerate the airflow. There are two types of nozzles commonly used: one is a tapered nozzle, and the other is a Laval nozzle. To analyze the flow phenomenon in nozzle is vital before transferring prototype design into industrial production, whether by laboratory experiment or computational simulation. In this paper, two different numerical methods are adopted to simulate gas behavior inside a simplified nozzle with upper and bottom symmetrical bumps. The first is to solve one dimensional Euler equations, and the other is to solve a scalar variable named velocity potential with small disturbance equations (SDE). The solutions under various inlet Mach numbers are compared by analysing the velocity fields and Mach number contours obtained by these two approaches. Similarities and differences between the Euler method and the potential SDE method for subsonic flow and supersonic flow are the key emphases in this work. For either subsonic or supersonic flow, the Mach number distribution along the nozzle's center line shows a consistent trend for both methods. In contrast, the values of maximum or minimum Mach number have corresponding differences. Moreover, by using potential SDE simulation, several types of shock waves are successfully captured. All results show that the incoming airflow decelerates at the leading edge of the nozzle, then accelerates when passing through the bumps, and finally decelerates back to the speed of inlet flow. The difference is that flows with varied Mach numbers has distinct velocity distributions in the nozzle.

1. Introduction
Computational fluid dynamics (CFD) is a cross discipline between mathematics, fluid mechanics, and computer programming aiming to simulate and analyse problems relating to fluid flows through numerical methods. With the development of computational science, CFD has been increasingly widely used, serving traditional fluid mechanics and engineering fields, such as aviation, aerospace, shipbuilding, and water conservancy. In aerospace engineering, the nozzle is a vital part of a rocket engine designed to accelerate the airflow by changing the geometry of its inner walls. Normally, two types of nozzles are commonly used in thermal power generation: one is the tapered nozzle, and the
other is the Laval nozzle [1]. The use of CFD to perform numerical simulation instead of expensive and
time-consuming ignition experiments has been an indispensable part of nozzle design. Thus the research
on nozzle numerical simulation is of great significance.

There are many numerical methods used to solve the flow phenomenon in a nozzle. Solving Euler
equations is one of the most widely used methods in many inviscid flow studies in open literatures.
Meanwhile, the potential theory is another powerful tool to analyse inviscid and irrotational flow in
many aviation engineering problems. Sterck and Rostrup [2] pointed out that the Euler equation of
stationary form is not easy to solve in transonic flow. But with the combination of adaptive integration
and the Newton method, they successfully solved the stationary form of the one-dimensional Euler
equation, and results demonstrate that the novel numerical method is fast and accurate enough. Colonna
et al. [3] compared two numerical methods for solving Euler equations and developed a new approach
for solving the one-dimensional nozzle flow. In this way, the stability problems caused by discontinuities
in transonic flows are avoided. Balakrishnan [4] presented an analytical solution of the potential
equation under Kutta-Joukowsky boundary conditions based on small disturbance theory. Liu et al. [5]
proposed a small disturbance linear stability analysis method for transonic potential flow and discussed
the relationship between the stability and convergence of iterative methods for solving the potential
equation.

In this paper, we considered a simplified model of the Laval nozzle and introduced different
numerical approximation algorithms for steady two-dimensional potential flow and unsteady one-
dimensional Euler flow of isentropic gas. This work aims to find similarities and differences in nozzle
flow using the potential method and Euler method. The full potential equation was first solved with
small disturbance approximation using the Gauss-Seidel iteration method. The Euler equations were
then solved using the Lax Friedrich finite difference method [6] for comparison. In both approaches, the
nozzle inlet Mach number was set as the only input variable, while velocity fields and Mach contours
were the outputs we expected to obtain.

1.1. The full velocity potential equation
In this subsection, the full potential equations are introduced. When dealing with a compressible fluid
such as air, there is a simpler way to obtain the fluid properties in a fluid domain other than solving
Navier–Stokes equations (NS equation). This effective tool is called as potential velocity equation. The
velocity potential equation has only one variable, which contains the entire property parameters in the
flow, for example, density, velocity, pressure, and temperature. This unique characteristic of the
potential flow equation implies that the continuity, momentum, and energy equations are combined in
an equation of velocity potential $\Phi$, a scalar variable. This velocity potential $\Phi$ is the only parameter
required to be solved. The equation of $\Phi$ is a second order partial differential equation like the NS
equation but with only one variable. Once the solution of $\Phi$ is obtained, then the velocity field is
calculated as the gradient of $\Phi$, which can be expressed as

$$ V \equiv \nabla \Phi $$

(1)

where $\Phi = \Phi(x, y, z)$ is the velocity potential in three dimensional, which is a scalar variable and $\nabla$ is
the nabla operator and $\nabla = \frac{\partial}{\partial x} \hat{i} + \frac{\partial}{\partial y} \hat{j} + \frac{\partial}{\partial z} \hat{k}$. Therefore,

$$ \nabla \Phi = \frac{\partial \Phi}{\partial x} \hat{i} + \frac{\partial \Phi}{\partial y} \hat{j} + \frac{\partial \Phi}{\partial z} \hat{k} $$

(2)

where $\hat{i}, \hat{j}, \hat{k}$ are standard unit vectors of $x, y$ and $z$ directions.

In this work, we must introduce several assumptions before using this powerful tool to analyze nozzle
flow. Note that not all flow problems have a corresponding function of velocity potential $\Phi$, and the first
assumption is that the flow is irrotational. Strictly speaking, there is not completely irrotational fluid
flow. But for many cases, such as air flows through a two-dimensional nozzle, we can approximate the
flow as irrotational, and this approximation conforms to engineering practice. The second hypothesis is
that the flow is treated as steady, inviscid, and isotropic. Based on the above two assumptions, a velocity
potential equation applicable to the 2-D nozzle flow can be derived by starting from the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{V}) = 0 \quad (3)$$

The steady state condition eliminates the time dependent term from the equation, and we get \(\nabla \cdot (\rho \mathbf{V}) = 0\). By expanding the equation and introducing the velocity potential \(\Phi\), the original continuity equation can be transformed as:

$$\rho \left( \frac{\partial^2 \Phi}{\partial x^2} + \frac{\partial^2 \Phi}{\partial y^2} + \frac{\partial^2 \Phi}{\partial z^2} \right) + \frac{\partial \Phi}{\partial x} \frac{\partial \rho}{\partial x} + \frac{\partial \Phi}{\partial y} \frac{\partial \rho}{\partial y} + \frac{\partial \Phi}{\partial z} \frac{\partial \rho}{\partial z} = 0 \quad (4)$$

Note that the above formula still contains two unknown variables of density \(\rho\) and \(\Phi\). We still need to eliminate the density \(\rho\) to achieve the final velocity potential equation.

Through a special form of Euler's momentum equation, Anderson [7] provides a method to eliminate the density term. Under the assumption of inviscidity and time independent, the NS equation degrades to Euler equation \(\rho(\mathbf{V} \cdot \nabla)\mathbf{V} = -\nabla p\). Moreover, the irrotational condition provides a relation that \(\nabla \times \mathbf{V} = 0\). By combining these two equations together, a special expression of the Euler equation can be derived as:

$$dp = -\rho \mathbf{V} \cdot d\mathbf{V} = -\rho d \left( \frac{\mathbf{V} \cdot \mathbf{V}}{2} \right) \quad (5)$$

Next, we are trying to eliminate the newly introduced variable pressure \(p\). Obviously, the relationship between pressure and density can be found by the sound speed formula \(a^2 = \left(\frac{\partial p}{\partial \rho}\right)_s\). Specifically, under isentropic condition, we have \(\frac{dp}{d\rho} = \left(\frac{\partial p}{\partial \rho}\right)_s\), thus we can express \(dp\) as \(dp = a^2 d\rho\) and the Euler equation is further simplified to:

$$d\rho = -\frac{\rho}{a^2} d \left( \frac{\left(\frac{\partial \Phi}{\partial x}\right)^2 + \left(\frac{\partial \Phi}{\partial y}\right)^2 + \left(\frac{\partial \Phi}{\partial z}\right)^2}{2} \right) \quad (6)$$

Now by combining the Eq. (4) and Eq. (6) and eliminating variable \(\rho\), the final velocity potential equation is obtained:

$$\left(1 - \frac{\Phi_x^2}{a^2}\right) \Phi_{xx} + \left(1 - \frac{\Phi_y^2}{a^2}\right) \Phi_{yy} + \left(1 - \frac{\Phi_z^2}{a^2}\right) \Phi_{zz} - \frac{2\Phi_x \Phi_y}{a^2} \Phi_{xy} - \frac{2\Phi_x \Phi_z}{a^2} \Phi_{xz} - \frac{2\Phi_y \Phi_z}{a^2} \Phi_{yz} = 0 \quad (7)$$

The stagnation sound speed \(a_0\) can be derived from the inlet velocity \(U_\infty\) and inlet Mach number \(M_\infty\) under the isentropic condition. Therefore, the local sound speed \(a\) has a thermodynamic relationship with \(a_0\):

$$\frac{a^2}{\gamma - 1} + \frac{V^2}{2} = \frac{a_0^2}{\gamma - 1} \quad (8)$$

Similarly, by introducing the velocity potential \(\Phi\), the above equation becomes:

$$a^2 = a_0^2 - \frac{\gamma - 1}{2} \left( \Phi_x^2 + \Phi_y^2 + \Phi_z^2 \right) \quad (9)$$

Therefore, the local sound speed is also expressed as a function of potential \(\Phi\), which means the full-speed potential equation has only one scaler variable \(\Phi(x, y, z)\). The above derivation forms a mathematical perspective that finding the solution of velocity potential \(\Phi\) is the core objective of the potential tool since the unique variable \(\Phi\) contains the complete information of flow field.

1.2. Simplification of full potential equation using SDE

Note that the derived full potential equation is suitable for any three-dimensional potential flow, and it has not specified a particular fluid domain. In this study, a simple 2-D flow is considered. That is, the
inlet flow has a constant horizontal velocity \( U_\infty \), whilst the vertical velocity of the inflow is 0. According to the definition of potential flow, the velocity potential of a uniform flow can be expressed as:

\[
\Phi = U_\infty \cdot x
\]  

(10)

when there is a small disturbance in the flow, the horizontal and vertical velocity has a corresponding small change compared with the uniform flow. If the small change of velocity is denoted as \( u' \) and \( v' \), SDE equation is then obtained as:

\[
U = U_\infty + u'
\]

(11)

\[
V = v'
\]

(12)

where \( U \) is the resulted horizontal velocity and \( V \) is the vertical velocity after the small perturbation. The small disturbance velocity \( u' \) and \( v' \) should also correspond to a potential function \( \phi \), which contains all the information about the flow disturbance. According to the definition of the velocity potential, the above SDE equation can be exchanged to a form of potential variable as:

\[
\frac{\partial \Phi}{\partial x} = U_\infty \cdot x + \frac{\partial \phi}{\partial x}
\]

(13)

\[
\frac{\partial \Phi}{\partial y} = \frac{\partial \phi}{\partial y}
\]

(14)

Then,

\[
\Phi(x, y) = U_\infty \cdot x + \phi(x, y)
\]

(15)

The above equation is used to substitute the potential variable of Eq. (7). Therefore, the full potential equation is expressed using the small disturbance potential \( \phi \) other than the original \( \Phi \). Anderson [7] demonstrated that when the inflow Mach number is less than 0.8 or greater than 5, the governing equation can be approximated as:

\[
\left(1 - M_\infty^2\right) \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0
\]

(16)

When the inflow Mach number is between 0.8 and 1.2 in the transonic region, the following equation needs to be used [7]:

\[
\left(1 + M_\infty^2 - (y + 1)M_\infty^2\right) \frac{\partial \phi}{\partial x} \frac{\partial^2 \phi}{\partial x^2} + \frac{\partial^2 \phi}{\partial y^2} = 0
\]

(17)

Obviously, Eq. (16) is linear while Eq. (17) is non-linear. For the case where the Mach number of inflow air is less than 0.8, the second-order central difference is used for variables \( x \) and \( y \) respectively. For the case where the inflow Mach number is greater than 1.2, the second-order upwind style is adopted in the horizontal direction. The second-order central difference is used in the vertical direction. Then the velocity field can be solved iteratively. When the flow is in the transonic region, the nature of the governing differential equation is between ellipse and hyperbolic, and it can no longer be solved by only one difference scheme. Alternatively, the local Mach number is calculated before iteration. When the local Mach number is less than 1, the governing equation exhibits the properties of an elliptic differential equation. Both of \( x \) and \( y \) need to adopt the central difference scheme. Otherwise, when the local Mach number is greater than 1, the governing equation exhibits hyperbolic properties, and the calculation uses time advanced difference format.

1.3. Discretization of the full potential equation with SDE

As mentioned above, there are two different discretization methods for subsonic flow and supersonic flow. Specifically, applying the finite difference theory for the subsonic flow, there is:

\[
\phi_{j,i} = \frac{\phi_{j+1,i} + \phi_{j-1,i} + (1 - M_\infty^2)\phi_{j,i+1} + (1 - M_\infty^2)\phi_{j,i-1}}{2(1 - M_\infty^2)} + 2
\]

(18)

Moreover, for the supersonic flow, there is:

\[
\phi_{j,i} = \frac{\phi_{j+1,i} + \phi_{j-1,i} - 2(1 - M_\infty^2)\phi_{j,i-1} + (1 - M_\infty^2)\phi_{j,i-2}}{2 - (1 - M_\infty^2)}
\]

(19)

It is noted that the above discretization format is only applicable to the internal grid points of the calculation domain. For the bottom and top boundaries of the nozzle, the format requires further
processing. The term $\frac{\partial^2 \phi}{\partial y^2}$ in Eq.16 is initially discretized as $(\phi_{j+1,i} - \phi_{j-1,i}) - (\frac{\partial \phi}{\partial y})_{\text{boundary}} \cdot \Delta x$.

Assuming that the nozzle is placed horizontally, the upper and lower boundary conforms to a shape function of $f(x)$, where $x$ is the horizontal length, the relationship between the vertical velocity at boundary and nozzle shape can be expressed as $(\frac{\partial \phi}{\partial y})_{\text{boundary}} = f'(x) \cdot \Delta x$, where $f'(x)$ is the first derivative of $f(x)$ and $\Delta x$ is the length of space discretization in $x$ direction. Through this conversion, the discretization format of the upper and lower boundary can be constructed. As for Eq.17, the discretization format needs to be selected carefully. The basic logic for selecting the format is as follows:

First, it is necessary to calculate $(1 + M_{\infty}^2 - (\gamma + 1)M_{\infty}^2 \frac{\partial \phi}{\partial x})$ for each point in the flow domain. If it is less than 1, the difference scheme for that specific point should be Eq.18. Otherwise, the difference scheme should be the format as shown in Eq.19.

### 1.4. The Euler equation

Euler equation is widely used in inviscid fluid dynamics obtained by applying Newton's second law to inviscid fluid. The definition of inviscid flow is a flow that ignores dissipation, viscous transport, mass diffusion, and heat conduction. Euler approach in this study is a 1-D problem and time dependent. Starting with the derivation of the governing equations of a model with constant cross section, the continuity, momentum, and energy equation can be listed as:

$$\frac{\partial}{\partial t} \left[ \rho \right] + \frac{\partial}{\partial x} \left[ \rho u \right] + \frac{\partial}{\partial x} \left[ \rho u^2 + p \right] = 0 \tag{20}$$

where $\rho$ is density, $u$ is the flow speed along $x$ axis, $p$ is pressure and $E$ is the total energy per mass unit and can be obtained by adding internal energy $e$ and kinematic energy together. The expression of $E$ can be written as $E = e + \frac{1}{2} u^2$. However, the actual shape of the nozzle, as shown in Fig.1, tells the fact that it does not have a constant cross-section area, thus the governing equation needs to be modified. In the following analysis, we introduced a new variable $A$ that represents the cross-sectional area of the nozzle, which is a function of $x$ [2] and $A'$ is the derivative of $A$ with respect to $x$, where $A$ is defined as:

$$A(x) = \pi r^2 = \pi \left[ \frac{1 - 2f(x)}{2} \right]^2 \tag{21}$$

![Cross-section area A(x)](image)

As shown in Fig.2, given a uniform flow in a tube of cross section $A(x)$, the continuity equation in integral form can be simplified as:

$$\rho_1 u_1 A_1 = \rho_2 u_2 A_2 \tag{22}$$

The corresponding differential equations is:

$$\frac{d}{dx} (\rho u A) = 0 \tag{23}$$

For unsteady flow, the time derivative is added to the formular Eq.23, so:

$$\frac{\partial (\rho A)}{\partial t} + \frac{\partial}{\partial x} (\rho u A) = 0 \tag{24}$$
As shown in the Figure 3, the momentum equation for steady flow in integral form can be simplified as:

\[
\frac{\partial}{\partial t} \int_{1}^{2} (\rho u A) \, dx + (\rho u A) \int_{1}^{2} \rho u A \, dx = (p_1 A_1 - p_2 A_2) + \int_{1}^{2} p \, dA
\]  

(25)

The corresponding differential equations is:

\[
\frac{\partial}{\partial t} (\rho u A) + \frac{\partial}{\partial x} (\rho u^2 A) = -A \frac{\partial p}{\partial x} - \frac{\partial}{\partial x} (pA) + \frac{\partial A}{\partial x}
\]  

(26)

The three equations are written in the form of a matrix with three vector variables of \(U\), \(F\), and \(G\) and one scalar variable of \(A\), the final expression of 1-D Euler equation for nozzle flow is obtained in Eq.29:

\[
U = \begin{bmatrix} \rho u \\ \rho u^2 + p \end{bmatrix}, F = \begin{bmatrix} \rho u^2 + p \\ p(\rho u + p) \end{bmatrix}, G = \begin{bmatrix} \rho u^2 \\ \rho E u + p u \end{bmatrix}
\]  

(27)

\[
E = \frac{p}{\rho (\gamma - 1)} + \frac{1}{2} u^2
\]  

(28)

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

(29)

The analytical solution of Eq.29 is hard to be obtained, the finite difference method was used to approximate the solution numerically. Specifically, the Lax Friedrich difference scheme was used to discretise the Eq.29:

\[
\frac{1}{\Delta t} \left[ U_j^{n+1} - \frac{1}{2} \left( U_{j-1}^{n} + U_{j+1}^{n} \right) \right] + \frac{1}{2\Delta x} \left[ F_j^{n+1} - F_{j-1}^{n} \right] + \frac{\Lambda'(j)}{A(j)} \frac{G_{j+1}^{n} + G_{j-1}^{n}}{2} = 0
\]  

(30)
The desired variable $U_{j+1}$ was moved to the left side of the equation so Eq.30 could be written as:

$$U_{j+1}^{n+1} = -\Delta t \frac{A'(j)}{A(j)} \left( \frac{G_{j+1}^n + G_{j-1}^n}{2} - \frac{\Delta t}{2\Delta x} \left[ F_{j+1}^n - F_{j-1}^n \right] \right) + \frac{U_{j+1}^n + U_{j-1}^n}{2}$$ (31)

Basic procedures for solving Eq.18, Eq.19 and Eq.31 are shown in Fig.4. For potential SDE method uses the Gauss-Seidel algorithm to solve the potential $\phi_{j,i}$. The end condition is to compare the difference of $\phi_{j,i}$ between any two adjacent iterations. The result can be seen as convergent when the difference goes to zero with the increasing number of iterations. For the Euler method, the result could be considered as convergent when the calculated value tends to be finite instead of blowing up as the number of time steps increases.

Like the 2-D problem in the potential method, the internal points were calculated first. And then boundary conditions are applied at the nozzle inlet and outlet. By analysing the eigenvalues of Eq.29, Different strategies were used to implement boundary conditions. The eigenvalues are calculated as $u$, $u+c$ and $u-c$. For subsonic flows $u<c$, there are two positive and one negative eigenvalues. In this case, two physical boundary conditions are needed at the inflow and one physical boundary condition at the outflow. For supersonic flow $u>c$, all eigenvalues are positive, and three physical boundary conditions are required at the inflow and none at the outflow.
2. Results and discussion

In this study, we considered a simple nozzle flow as shown in Fig. 5, which is a sketch of this model with an initial uniform flow of $U_\infty$ at its inlet. The flow domain has a length of 3 unit in $x$ axis and a width of 1 unit in $y$ axis. Two small bumps with symmetry shapes are generated in the interval of $1 \leq x \leq 2$ at the bottom and top surfaces, respectively.

![Figure 5. Computational domain of simple nozzle model](image)

2.1. Results of potential method

2.1.1 Linear results in subsonic and supersonic flow

Fig. 6 shows the velocity field and Mach number contour in a subsonic flow where the inflow Mach number is 0.7. It can be seen that when the incoming flow comes to the leading edge and trailing edge of the small bump, the Mach number decreases from around 0.70 to 0.55. While passing through the tapered channel, the flow velocity increases and reaches the maximum Mach number of 0.825 at the throat. Overall, the result shows a symmetry characteristic with the vertical centerline of the throat.

![Figure 6. The velocity field and Mach number contour with inflow Mach number being 0.7](image)
Different from the result obtained by the subsonic flow that exhibits a horizontal symmetry characteristic, the result of supersonic flow does not have this feature. Fig.7 demonstrates the velocity field and Mach number contour when inflow Mach number is 1.5.

It can be seen that although the streamline diagram has similarities compared with the subsonic flow, the Mach number contour is completely different. When the incoming flow is supersonic, two oblique shock waves are formed at the leading and trailing edges of the bumps. When the air flows above the bump surface, it first slows down rapidly at the location where the front shock wave occurs, then accelerates at the throat, and finally passes through a shock wave again at the trailing edge. In order to compare results with the 1-D Euler method, the Mach number distribution along the horizontal centerline is extracted from the Mach contour and is plotted in Fig.8.

---

**Figure 7.** The velocity field and Mach number contour with inflow Mach number being 1.5

---

**Figure 8.** Mach number along the horizontal center line with inlet Mach number being 1.5
2.1.2 Nonlinear results in transonic flow

When the incoming flow Mach number is in the transonic region where $0.8 \leq Mach \leq 1.2$, the solution is more complicated and needs to be discussed separately. On the one hand, if the inlet uniform flow has a Mach number of less than 1, the subsonic inflow will accelerate as it passes through the throat. If it accelerates to a speed of supersonic flow, a shock wave will be generated. Fig.9 shows the case where a normal shock wave appears near the throat. When the air flows through the shock wave, the supersonic air suddenly reduces its speed to subsonic and then flows out of the computational domain. For comparison with Euler results, Fig.10 shows the distribution of Mach numbers at its horizontal centerline.

Figure 9. The velocity field and Mach number contour with inflow Mach number being 0.9

Figure 10. Mach number along the horizontal center line with inlet Mach number being 0.9
On the other hand, when the incoming flow is supersonic, as analysed in the previous section, the supersonic flow in a transonic zone also results in a shock wave around the leading edge of the bump. It decelerates to subsonic, as shown in Fig.11. When passing through the throat, the air accelerates to supersonic again. It produces shock waves near the trailing edge, resulting in a significant speed reduction, and finally, the air flows out with the same Mach number as the incoming flow. Also, the Mach number distribution along the nozzle centerline is given in Fig.12 for comparison.

![Figure 11. Velocity and Mach number with inlet Mach number being 1.2](image1)

![Figure 12. Mach number along the horizontal center line with inlet Mach number being 1.2](image2)

2.2. Results of Euler method and comparison

As the Euler equation in this work is only considered in 1-D, we could only obtain the speed of air and its Mach number distribution along the center line of the nozzle. Mach number distribution, as shown
in Fig.13, presents the result where calculation step time is 0.4 (200 times of iterations), and inlet Mach number is 0.7. The Mach number reduces before air hits the front edge of the nozzle and then substantially increases to 0.82 at the smallest cross-sectional area, then the air soon slows down back to the initial value of Mach number 0.7.

Figure 13. Mach number along horizontal center line at 200 iterations (Time Step 0.4) of Euler method with inlet Mach number being 0.7

Comparing the subsonic Euler result with the potential SDE result of 500 iterations (Fig.14), it could be found that the distribution of Mach number has a similar tendency along the horizontal centerline. However, it is easy to see a decrease of boundary Mach number for the potential SDE method from 0.70 to around 0.69 in Fig.14. This slight decrease in inlet and outlet Mach number might result from the disturbance of the bumps, which transports from edges of the bumps to the left and right boundaries of the computing domain when the flow is subsonic. This phenomenon could be one of the characteristics of elliptic partial differential equations.

Figure 14. The Mach number along the center line at potential SDE 500 iterations with inlet Mach number being 0.7
When the time step increases from 0.4 (200 iterations) to 20 (10,000 iterations) for the Euler method, and the inlet Mach number remains 0.7, the Mach number distribution along the centerline is given in Fig.15. Mach number maintains around 0.55 before air hits the humps. Then it rises to a maximum value of 0.78 at the throat. After passing the throat, as the cross-sectional area gradually increases, the Mach number drops down back to around 0.50. Again, there exists the phenomenon that the value of the Mach number at the boundaries of the computational domain is affected by the disturbance of bumps.

![Mach number distribution](image1.png)

**Figure 15.** Mach number along the center line at 10,000 iterations (Time Step 20) of Euler method with inlet Mach number being 0.7

The above Euler result (Fig.15) is compared to the result from the potential method at the same inlet condition as well. The result from the potential SDE method with 10,000 iterations is given in Fig.16, which again indicates the Mach distribution has a similar tendency as Euler does. Still, the potential result is symmetrical, and both the maximum and minimum of Mach numbers are slightly bigger than the Euler’s.

![Mach number distribution](image2.png)

**Figure 16.** Mach number along the center line at potential SDE 10,000 iterations with inlet Mach number being 0.7
When the inlet Mach number is added up to 0.9 in the transonic zone, the results for both the Euler method and potential SDE method show the characteristics of transonic flow in the nozzle. Fig.17 shows the Mach number distribution result calculated by the Euler method with an inlet Mach number of 0.9. The result shows that when the incoming flow is close to the leading edge, the Mach number reduces to a minimum of about 0.65. The air then accelerates in the nozzle and reaches the speed of sound near the position of the throat (x=1.5). Subsequently, the air flow does not decelerate. Still, it continues to accelerate with the expansion of the nozzle section until a shock wave appears, where the Mach number drops sharply and returns to the inlet Mach number 0.9 and finally flows out of the computation domain. As shown in Fig.18, the Mach number distribution illustrates the same characteristic as Euler does for potential SDE results. That is, the air speeds up to Mach number of 1.0 around the minimum cross-sectional area (x=1.5) and keeps accelerating in the expansion section until a normal shock wave occurs, as presented in section 2.1.2. However, although the shape of the Mach number distribution curves of both methods is similar, the value is slightly different. The highest and lowest Mach numbers calculated by the potential SDE method are both about 0.1 higher than those of the Euler method. The shock wave captured by the potential SDE method has a much larger attenuation rate, which can better reflect the non-continues characteristics of the shock waves in flow dynamics.

![Mach number distribution](image1.png)

Figure 17. Mach number distribution along the horizontal center line of Euler method at 500 iterations with inflow Mach number being 0.9

![Mach number distribution](image2.png)

Figure 18. Mach number along the horizontal center line of potential SDE method at 500 iterations with inflow Mach number being 0.9
The results of supersonic flow at a Mach number of 1.5 are compared in Fig 19 and Fig.20 for both methods. It can be seen that the shape of the Mach number distribution curves is still similar, but the shock wave position has changed significantly. The shock wave position calculated by the Euler method is obviously closer to the leading edge than the potential SDE method, and the maximum and minimum values of the Mach number are also moderately different. It is worth noting that, unlike the characteristic of subsonic elliptic differential equations, the supersonic flow has the characteristic of hyperbolic differential equations. This feature is reflected in the results that the bump disturbance information in the supersonic flow will not be transmitted to the incoming flow, which corresponds to the fact that the values of the Mach number at the inlet boundary remain stable at 1.5 and will not drop slightly as Fig.14 does.

Figure 19. Mach number distribution along the center line of Euler method at 500 iterations with inflow Mach number being 1.5

Figure 20. Mach number along the center line of potential SDE method at 500 iterations with inflow Mach number being 1.5
3. Conclusion and future works
In this paper, we presented numerical simulations in a nozzle with two different methods. The first approach uses the finite-difference method to solve the 2-D potential equation of the nozzle flow, while the second approach is to solve the Euler equations of the nozzle flow but only for the horizontal centerline of the nozzle, which could be seen as a 1-D problem. The biggest difference between the two methods is that the former introduces a scalar variable named velocity potential. The velocity potential is first solved, and then the velocity field is obtained by calculating the gradient of the potential. This is a time-independent method. In contrast, the latter directly obtain the time-dependent velocity field by solving the Euler equations, which is a set of equations derived from Navier-Stokes equations after the inviscid hypothesis. For the 1-D nozzle flow, the set of Euler equations contain three individual variables and three equations of conservation of mass, momentum, and energy.

The similarities and differences of results are compared between the potential SDE and Euler method. Two methods numerically simulate both subsonic flow and supersonic flow. For subsonic or supersonic flow, the Mach number distribution of the two methods along the nozzle's center line shows a consistent trend. That is, the airflow decelerates at the leading edge of the nozzle, accelerates at the throat, and finally decelerates at the trailing edge. We successfully captured the shock waves in the nozzle by non-linear simulation of transonic flow in the potential SDE simulation, including leading edge shock waves, trailing edge shock waves, and normal shock waves. Since the potential flow method is a two-dimensional simulation, the obtained velocity field, Mach contour, shape, and position of the shock waves can be clearly displayed. This is the capability that the one-dimensional Euler method does not have. While in the Euler method, the changes of velocity and Mach number with time on the centerline can be obtained due to its time-dependent feature.

Some future work could be done to refine the simulation. Firstly, the potential flow method can be upgraded from two-dimensional to three-dimensional, while the Euler method can be upgraded from one-dimensional to three-dimensional. For the Euler method, different discretisation schemes can be explored other than the Lax Friedrich scheme. Furthermore, viscosity can be added to the numerical calculation, which means the results must be obtained by directly solving the NS equation or solving a variant of the NS equation with turbulence models, such as Reynolds Averaged NS equation or Large Eddy Simulation.

Overall, this study numerically simulates the air flow in a simplified nozzle with symmetrical upper and lower walls and obtains the velocity distribution in the nozzle at different inlet Mach numbers. In the foreseeable future, with the rapid development of aerospace industry, there will still be a tendency of replacing expensive and time-consuming wind tunnel experiments with numerical simulations. This work provides a feasible example with the aid of CFD for nozzle design and airfoil optimization in the aerospace industry. On this basis, researcher can further analyze the flow phenomenon in a nozzle by optimizing the wall function, altering numerical simulation methods and discrete scheme.

References
[1] PANG, S.M. and CHEN, P.M., 2011. Taper Nozzle Design on CFD [J]. Machine Building & Automation, 1.
[2] De Sterck, H. and Rostrup, S., 2007 Direct Numerical Solution of the Steady 1D Compressible Euler Equations for Transonic Flow Profiles with Shocks.
[3] Colonna, G., Tuttafesta, M. and Giordano, D., 2001. Numerical methods to solve Euler equations in one-dimensional steady nozzle flow. Computer physics communications, 138(3), pp.213-221.
[4] Balakrishnan, A.V., 2004. Transonic small disturbance potential equation. AIAA journal, 42(6), pp.1081-1088.
[5] Liu, Y., Luo, S. and Liu, F., 2017. Multiple solutions and stability of the steady transonic small-disturbance equation. Theoretical and Applied Mechanics Letters, 7(5), pp.292-300.
[6] Kröner, D. and Thanh, M.D., 2005. Numerical solutions to compressible flows in a nozzle with variable cross-section. SIAM journal on numerical analysis, 43(2), pp.796-824.
[7] Anderson, J.D., 2003. Modern compressible flow. Tata McGraw-Hill Education.