Euler and Small Disturbance Equation-based Simulation of a Nozzle Flow

Kuihao Huang*
Department of Mechanical Engineering, University of California Santa Barbara, Santa Barbara, 93106, United States
*Kuihaohuang@ucsb.edu

Abstract: The de Laval nozzle has been widely studied and used in many industries ranging from the aerospace to the dairy industry. This study investigated two numerical methods used to simulate nozzle flow, namely one-dimensional Euler equations and the small disturbance equations (SDE) based simulation. The simulation results reveal that the SDE method accurately captured the shock bubbles in the nozzle flow. However, the one-dimensional Euler method is not sufficient to handle two-dimensional nozzle flow simulation because at any particular nozzle location, a mix of subsonic and supersonic flow can occur. This research allows an informed choice between Euler and SDE method.

1. Introduction
Euler and the small disturbance equation (SDE) method are both ways to simulate fluid problems numerically, which are widely used in the computational fluid dynamics (CFD) [1]. Euler equations are a simplified version of Navier-Stokes equations by excluding the effect of the viscosity on the fluid, also known as inviscid flow, which only describes the fluid using velocity, pressure, and density. SDE is a simplified form of the full potential equations that is intended to be used in situations where there are thin obstacles, such as airfoils. Euler is a one-dimensional method to solve a three-dimensional problem, and the current paper's version of the SDE is a two-dimensional method. Euler and the SDE method have been widely used in the numerical simulation, especially for the nozzle flow. For instance, Colonna et al. [2] used numerical method, namely Euler equations to solve one-dimensional and steady nozzle flow. The result of the simulation was successful. However, they drew the conclusion that the simulation had numerical errors and discontinuity in the transonic region was the main difficulty when using the Euler scheme. Moreover, Van et al. [3] used Low Frequency Small-Disturbance (LF-TSD) equation to solve fluid-structure interaction (FSI) problems. They concluded that LF-TSD was a simple way to simulate the FSI problem while providing rich solutions. Their method is transient, but they use it at a steady state.

The objective of this paper is to simulate nozzle flow using both Euler and the 2D steady SDE method, and compare their results in terms of Mach number, velocity, and pressure. The paper is organized as follows. In Section 2, the methods used to simulate the nozzle flow are presented along with equations. Section 3 will discuss the results, findings, then followed by the conclusions in Section 4.
2. Methods

2.1 Euler equations

The first simulation method is using 1D inviscid Euler equations [4] [5]. The 1D Euler equation in the non-conservative form needs to be introduced. The primitive variables are:

\[ U = \begin{pmatrix} \rho \\ u \\ p \end{pmatrix} = \begin{pmatrix} \frac{m}{\rho} \\ (y - 1) \left( \epsilon - \frac{m^2}{2\rho} \right) \end{pmatrix} \]  

(1)

And the corresponding equations are:

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

\[ \frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + \frac{1}{\rho} \frac{\partial p}{\partial x} = 0 \]  

(2)

\[ \frac{\partial p}{\partial t} + u \frac{\partial p}{\partial x} + \rho c^2 u_x = 0 \]

In short, the above equation can be denoted as:

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

(3)

where the matrix \( A \) is given by:

\[ A = \begin{pmatrix} u & \rho & 0 \\ 0 & u & \frac{1}{\rho} \\ 0 & \rho c^2 & u \end{pmatrix} \]  

(4)

To discover the characteristic variables, a diagonalization procedure needs to be performed on matrix \( A \) to find the eigenvalues and eigenvectors. The eigenvalues are later used to determine different boundary conditions. This gives:

\[ \det (A - \lambda I) = (u - \lambda) [(u - \lambda)^2 - c^2] = 0 \]  

(5)

The resultant root from Eq. (5) are eigenvalues with their corresponding eigenvectors \( \lambda_1 = u, \lambda_2 = u + c, \) and \( \lambda_3 = u - c, \) with

\[ \ell_1 = \begin{pmatrix} 1 \\ 0 \\ 0 \end{pmatrix}, \ell_2 = \begin{pmatrix} \rho \\ \frac{1}{2c} \\ \frac{1}{2} \end{pmatrix}, \ell_3 = \begin{pmatrix} -\rho \\ \frac{1}{2c} \\ -\frac{1}{2} \end{pmatrix} \]

Since the flow is going through a nozzle, the nozzle flow equation is used here, which is adding source terms depending on the nozzle shape on the right-hand side of the Euler equation. Assuming the cross-section of the nozzle is \( S(x) \), then the equations are expressed as:

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

\[ \frac{\partial (\rho u S)}{\partial t} + \frac{\partial (\rho u^2 + p) S}{\partial x} = p \frac{dS}{dx} \]  

(6)

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

By using the primitive variables, the above equations can be rearranged as:
\[
\frac{\partial \rho}{\partial t} + u \frac{\partial \rho}{\partial x} + \rho \frac{\partial u}{\partial x} = -\frac{\rho u}{S} \frac{dS}{dx}
\]
\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + \frac{1}{\rho} \frac{\partial p}{\partial x} = 0
\]
\[
\frac{\partial p}{\partial t} + u \frac{\partial p}{\partial x} + \gamma p \frac{\partial u}{\partial x} = -\frac{\gamma \rho u}{S} \frac{dS}{dx}
\]

and \(c^2 = \gamma RT, p = \rho RT,\) and \(\frac{ds}{dx}\) is evaluated analytically.

Eq. (7) are a set of equations used to describe the nozzle flow, and the position component of the equation can be discretized with a central difference [6] for the subsonic case and upwind method [7] (backward difference for flow to the right, forward for flow to the left) for the supersonic case. Then MATLAB’s built-in solver ODE23tb [8] is used to integrate the equation in time. The discretization schemes for the subsonic and supersonic case are expressed as Eq. (8) and (9) respectively:

1. **Subsonic case:**

   \[
   \frac{\partial \rho_i}{\partial t} + u_i \frac{p_{i+1} - p_{i-1}}{2\Delta x} + \rho_i \frac{u_{i+1} - u_{i-1}}{2\Delta x} = -\frac{\rho_i u_i}{S_i} \frac{dS}{dx}_i
   \]

   \[
   \frac{\partial u_i}{\partial t} + u_i \frac{u_{i+1} - u_{i-1}}{2\Delta x} + \frac{1}{\rho_i} \frac{p_{i+1} - p_{i-1}}{2\Delta x} = 0
   \]

   \[
   \frac{\partial p_i}{\partial t} + u_i \frac{p_{i+1} - p_{i-1}}{2\Delta x} + \gamma p_i \frac{u_{i+1} - u_{i-1}}{2\Delta x} = -\frac{\gamma \rho_i u_i}{S_i} \frac{dS}{dx}_i
   \]

2. **Supersonic case with flow left to right (only this case is shown here for example, but right to left flow is also implemented in the code):**

   \[
   \frac{\partial \rho_i}{\partial t} + u_i \frac{p_{i} - p_{i-1}}{\Delta x} + \rho_i \frac{u_{i} - u_{i-1}}{\Delta x} = -\frac{\rho_i u_i}{S_i} \frac{dS}{dx}_i
   \]

   \[
   \frac{\partial u_i}{\partial t} + u_i \frac{u_{i} - u_{i-1}}{\Delta x} + \frac{1}{\rho_i} \frac{p_{i} - p_{i-1}}{\Delta x} = 0
   \]

   \[
   \frac{\partial p_i}{\partial t} + u_i \frac{p_{i} - p_{i-1}}{\Delta x} + \gamma p_i \frac{u_{i} - u_{i-1}}{\Delta x} = -\frac{\gamma u_i p_i}{S_i} \frac{dS}{dx}_i
   \]

After taking care of interior points in nozzle flow, physical and numerical boundary conditions need to be considered. The goal is to supply a different boundary condition for points at the boundaries that are consistent with the nozzle flow governing equations. Boundary conditions for inflow and outflow are shown below:

1. **Inflow boundary condition:** For the inflow, the information is moving from left to right in this case. Two physical boundary conditions and one numerical boundary condition are needed because there are two positive characteristics coming in on the left and one negative characteristic leaving for a subsonic inlet.

   a. Physical boundaries are specified as: \(\rho = \rho_\infty, u = U_\infty.\)

   b. A numerical boundary is obtained by an approximation of Eq. (7) with forward differencing,

   \[
   \frac{\partial \rho_i}{\partial t} + u_i \frac{p_{i+1} - p_{i}}{\Delta x} + \rho c^2 \frac{u_{i+1} - u_i}{\Delta x} = -\frac{\rho u c^2}{S} \frac{dS}{dx}
   \]

2. **Outflow boundary condition:** There are two possible boundary conditions for the outflow, as it depends on if the flow is subsonic or supersonic.

   a. For subsonic where the flow velocity is lower than the speed of sound then \(u, u + c\) are positive and \(u - c\) is negative. We have two numerical boundary conditions using backward differencing to Eq. (7) which is in the same fashion as Eq. (9). And, \(u - c\) could be a physical boundary condition by specifying a backpressure at the outlet, however, we choose to extrapolate from the interior points.
b. For supersonic where the flow velocity is greater than the speed of sound then $u, u + c,$ and $u - c$ are all positive. All of them will be numerical boundary conditions by using backward differencing to Eq. (7) as the flow is leaving the domain, so only the information inside the domain can be used.

2.2 Small disturbance method starting with full potential equation

To understand the derivation of the transonic small disturbance equation, one needs to start with the simplifications of the Euler equations, which will become the full potential equation. The simplification starts with additional assumptions: the flow is steady, irrotational, and isentropic. Then the Euler equations can be combined into a single equation called the velocity potential equation. One can start with writing the steady continuity equation in a Cartesian coordinate system [9] [10]:

$$\frac{\partial \rho u}{\partial x} + \frac{\partial \rho v}{\partial y} + \frac{\partial \rho w}{\partial z} = 0 \quad (11)$$

Where $\rho, u, v, w$ denote density, $x$-direction velocity, $y$-direction velocity and $z$-direction velocity respectively.

Then by replacing the velocity components with:

$$u = \frac{\partial \Phi}{\partial x}, v = \frac{\partial \Phi}{\partial y}, w = \frac{\partial \Phi}{\partial z}$$

The continuity equation becomes:

$$\frac{\partial}{\partial x}(\rho \Phi_x) + \frac{\partial}{\partial y}(\rho \Phi_y) + \frac{\partial}{\partial z}(\rho \Phi_z) = 0 \quad (12)$$

The momentum equation with the assumptions of steady, isentropic and irrotational flow can be written as:

$$\frac{V^2}{2} + \int \frac{dp}{\rho} = \text{const} \quad (13)$$

Where $V, \rho, p$ denotes velocity, density and pressure respectively.

Writing the momentum equation in differential form:

$$dp = -\rho \left( \frac{V^2}{2} \right) = -\rho \left( \frac{\Phi_x^2 + \Phi_y^2 + \Phi_z^2}{2} \right) \quad (14)$$

The speed of sound for constant entropy process combined with perfect gas equation is given by:

$$a = \sqrt{\frac{\gamma p}{\rho}} = \sqrt{\gamma RT}$$

By combining the speed of sound equation and the momentum equation, the following equation can be obtained:

$$dp = -\frac{\rho}{a^2} \left( \frac{\Phi_x^2 + \Phi_y^2 + \Phi_z^2}{2} \right) \quad (15)$$

The velocity potential equation can be obtained by substituting expressions for the density term in $x, y,$ and $z$ direction.

$$\left( 1 - \frac{\Phi_x^2}{a^2} \right) \phi_{xx} + \left( 1 - \frac{\Phi_y^2}{a^2} \right) \phi_{yy} + \left( 1 - \frac{\Phi_z^2}{a^2} \right) \phi_{zz} - \frac{2 \Phi_x \Phi_y}{a^2} \phi_{xy} - \frac{2 \Phi_x \Phi_z}{a^2} \phi_{xz} - \frac{2 \Phi_y \Phi_z}{a^2} \phi_{yz} = 0 \quad (16)$$

A small disturbance equation can be obtained by further simplifying the velocity potential equation. It is done in the case of thin obstacles or a slender body, such as thin airfoils. A small-perturbation theory is used in this analysis as the free stream is only slightly disturbed by the thin obstacles. In the case of small disturbance approximation, we will restrict the flow in 2-D. Since the obstacles are considered small and their effect on the flow is small as well. The velocity component of the flow that is slightly disturbed can be written as:
\[ u = U_\infty + u' = \frac{\partial \Phi}{\partial x} \]
\[ v = v' = \frac{\partial \Phi}{\partial y} \]

where the prime denotes perturbation velocity. Letting \( \phi \) be the perturbation velocity potential, we get:

\[ u = \frac{\partial \Phi}{\partial x} = U_\infty + u' = U_\infty + \frac{\partial \phi}{\partial x} \]
\[ v = \frac{\partial \Phi}{\partial y} = U_\infty + \frac{\partial \phi}{\partial y} \]

The compressible Bernoulli equation is given by

\[
\frac{V^2}{2} + \frac{p}{\gamma - 1} = \text{const}
\]

By substituting the perturbation velocity along with the speed of sound equation into compressible Bernoulli equation, the following equation can be obtained:

\[
a^2 = a_\infty^2 - \frac{\gamma - 1}{2} [2u'U_\infty + (u')^2 + (v')^2]
\]

Since the flow is only slightly disturbed, we can make the assumption that

\[
\frac{u'}{U_\infty} \ll 1, \quad \frac{v'}{U_\infty} \ll 1
\]

Thus, the compressible Bernoulli equation becomes:

\[
a^2 = a_\infty^2 - (\gamma - 1)u'U_\infty
\]

Combining the compressible Bernoulli equation and the velocity potential equation yields:

\[
\left[ \frac{(1 - M^2)}{M_\infty^2} \right] - (\gamma - 1) \frac{u'}{U_\infty} M_\infty^2 \phi_{xx} + \phi_{yy} = 0
\]

In order to use the small disturbance equation to simulate nozzle flow different discretization methods need to be used because the subsonic and supersonic flow has different physics, so the numerical scheme also needs to be different to capture the phenomenon properly. For the subsonic case, the central differences approach is used to approximate the numerical scheme. For the supersonic case, an upwind scheme is used as the information can only travel downstream.

For the subsonic case where the Mach number (\( M \)) is less than 1 we have along with the assumption that \( h = \Delta x = \Delta y \) we have

\[
(1 - M^2)(\phi_{i+1,j} - 2\phi_{i,j} + \phi_{i-1,j}) + (\phi_{i,j+1} - 2\phi_{i,j} + \phi_{i,j-1})
\]

With the same square-grid assumption, for the supersonic case where the Mach number (\( M \)) is greater than 1 we must upwind because of the wave-equation-like behavior to maintain stability,

\[
(1 - M^2)(\phi_{i,j} - 2\phi_{i-1,j} + \phi_{i-2,j}) + (\phi_{i,j+1} - 2\phi_{i,j} + \phi_{i,j-1})
\]

Replacing \( (1 - M^2) \) as \( K \),

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

Combining the subsonic and supersonic cases into a single equation,

\[
\mu_i K_{i-1,j} (\phi_{i,j} - 2\phi_{i-1,j} + \phi_{i-2,j}) + (1 - \mu_i) K_{i,j} (\phi_{i+1,j} - 2\phi_{i,j} + \phi_{i-1,j}) + (\phi_{i,j+1} - 2\phi_{i,j} + \phi_{i,j-1}) = 0
\]

and when \( M < 1, \mu = 0 \) and \( M > 1, \mu = 1 \).

After deriving the equation used to simulate the nozzle flow for interior points which corresponding to Section 3 in Fig. 1, the boundary conditions will be considered. In this simulation, the flow is flowing
through a nozzle shape with solid surfaces at the top and bottom. The other two directions, namely inlet, and outlet are extending to infinity. In Fig. 1 there are in total of 6 sections that represent different regions of the nozzle and the boundary condition for each section will be discussed below. The equations are used as ghost points.

- **Section 1 (Nozzle inlet):** \( \phi_{i=1,j} = 0 \)
- **Section 2 (Nozzle Inlet):** \( \phi_{i-1,j} = 0 \)
- **Section 4 (Nozzle outflow):** 
  \[
  K_{i,j} = 1 - M_{\infty}^2 - (y + 1)M_{\infty}^2 \left( \frac{3\phi_{i,j} - 4\phi_{i-1,j} + \phi_{i-2,j}}{2h} \right)
  \]
- **Section 5 (Nozzle top surface):** \( \phi_{i,j+1} = \phi_{i,j-1} + 2hf'(x_i) \)
- **Section 6 (Nozzle bottom surface):** \( \phi_{i,j-1} = \phi_{i,j+1} + 2hf'(x_i) \)

![Fig. 1 Illustration of the discretization for the nozzle flow simulation](image)

3. Results and discussion

The simulation for both Euler and SDE method is based on a converging-diverging nozzle, shown in Figure 2, and the shape of the nozzle is approximated by the equation below,

\[
y = 2\beta(1 - x)x
\]

with \( \beta = 0.084 \)

The solution of the simulation is on a grid with \((x, y) \in [0; 3] \times [0; 3], \Delta x = 1/60.\)

![Fig. 2. Nozzle geometry with flow direction](image)

3.1 Results of Euler-based and SDE simulations along the centerline

The velocity, pressure, and Mach number obtained by the Euler-based simulation and the SDE-based simulation along the centerline are presented in Figures 3-5 respectively. It can be noted that the result from Euler and SDE method does not agree with each other. As the result from the SDE method is verified against another source (shown in the next subsection), we can conclude that the one-dimensional Euler method is not sufficient to simulate this particular two-dimension nozzle flow as there are supersonic and subsonic flow happening at the same location in the nozzle. The contour graphs shown...
in Fig. 6 obtained from the current SDE method shows such a scenario where the shock happens when the Mach number is greater than one and is represented by the thickened black line.

(a) Euler

(b) SDE

Fig. 3. The velocity plot obtained by the Euler-based simulation and the SDE-based simulation along the centerline

(a) Euler

(b) SDE

Fig. 4. The pressure plot obtained by the Euler-based simulation and the SDE-based simulation along the centerline
3.2 Results of the SDE-based simulation

Figure 6 shows flow around the neck of the nozzle, also known as biconvex airfoil with color bars. The dark blue contour indicates the position of the supersonic pocket, and it matches with the Optstal contour plot [3] which is indicated by white contour. It is a validation of the result of SDE method.
Figure 7 compares the bottom surface pressure of the SDE-based simulation result and that of the Opstal [3]. The two pressure plots are nearly identical except for different reference pressure, where there is a sudden jump in pressure or discontinuity which means a shock is happening at that location. The comparison further validated the simulation result from the SDE method of this paper.

4. Conclusions
In this paper, Euler-based and SDE-based numerical simulation methods are utilized to obtain the flow filed inside the de Laval nozzle. The results of the simulations reveal that the SDE method leads to accurate performance in simulating nozzle flow. However, the one-dimensional Euler method is not sufficient to handle two-dimensional nozzle flow simulations. While both the Euler-based and SDE-based numerical simulation methods neglect the viscosity effect (except for artificial viscosity in the Euler method), the Navier-Stokes based method could be adopted to find out the influence of the viscosity on the nozzle flow. Moreover, a detailed convergence study can be done to the simulation to check the error in the Euler and SDE method.

References
[1] Tannehill, J., Anderson, D., Pletcher, R. (1997) Computational Fluid Mechanics and Heat Transfer. Taylor&Francis, Washington, DC.
[2] Colonna, G., Tuttafesta, M., Giordano, D. (2001) Numerical methods to solve Euler equations in one-dimensional steady nozzle flow. Computer Physics Communications.
[3] Opstal, T., Hulshoff, S., Verhoosel, C. (2011) A Robust Solution Procedure for the Transonic Small-Disturbance Equation in Fluid-Structure Interactions. The open Aerospace Engineering Journal.
[4] Philip J. Pritchard. Introduction to Fluid Mechanics. Fox and McDonald’s.
[5] Nancy Hall, 2021. Euler Equations. https://www.grc.nasa.gov/www/k-12/airplane/eulereqs.html.
[6] Wikipedia. Finite Difference. https://en.wikipedia.org/wiki/Finite_difference
[7] Wikipedia. Upwind scheme. https://en.wikipedia.org/wiki/Upwind_scheme
[8] MathWorks, 2021. Solver. https://www.mathworks.com/help/simulink/gui/solver.html.
[9] Michael Giles. (1988) Non-Reflecting Boundary Conditions for the Euler Equations. MIT
[10] John D. Anderson Jr. (2007) Fundamentals of Aerodynamics. McGraw-Hill.