Research progress in numerical simulation of Laval nozzle

Congchi Huang\textsuperscript{1, a, †}, Tianyue Wang\textsuperscript{2, b, †} and Zongyu Zhang\textsuperscript{3, c, †}

\textsuperscript{1}Department of Mechanical Engineering, South China Agricultural University, Guangzhou, Guangdong, 510642, China;
\textsuperscript{2}Department of Mechatronic Engineering, Tianjin University of Technology, Tianjin, Tianjin, 300384, China;
\textsuperscript{3}Department of Vehicle Engineering, Yan Shan University, Qinhuangdao, Hebei, 066044, China.

\textsuperscript{a}hcc@stu.scau.edu.cn, \textsuperscript{b}2648622112@qq.com, \textsuperscript{c}zyysdx@stumail.ysu.edu.cn.

\textsuperscript{†}These authors contributed equally.

Abstract. The Laval nozzle is a very widely used device that has made great contributions due to its invention in aerospace, mechanical engineering, applied chemistry, etc. The development of many modern technologies relies on the establishment of nozzles. In this paper, the current research advances on the numerical simulation methods of the nozzle flow and nozzle-related investigation or applications in different fields are reviewed. For example, the analytical and numerical simulation of nozzles in subsonic and supersonic modes are mentioned in this article. In summary, although the applications for various fields of nozzles are now very mature, the solution of the Navier-Stokes equation for nozzles is still a future direction where science and mathematics need to be strongly developed.

1. Introduction

The de Laval nozzle [1] is a tube that is pinched in the middle so that it forms a carefully balanced asymmetrical hourglass shape, as illustrated in Figure 1. So, the first half of this shape is contracted from large to small toward the center to a narrow throat. The narrow throat is followed by another expansion from small to large outward to the bottom of the arrow. The gas in the arrow body flows under high pressure into the front half of the nozzle, passes through the narrow throat, and then escapes through the back half. This architecture allows the velocity of the airflow to vary depending on the spray cross-sectional area, allowing the airflow to go from subsonic to sonic and up to supersonic speeds. It is used to accelerate the hot pressurized gas passing through it in the axial direction to higher supersonic speeds, converting the thermal energy of the flow into kinetic energy. The nozzle is widely used in certain types of steam turbine and rocket engine nozzles [2]. It can also be found in supersonic jet engines [3].
Figure 1. Illustration of de Laval nozzle

Its operation relies on the different characteristics of gases flowing at subsonic, sonic, and supersonic speeds. If the tube is narrower due to constant mass flow, the subsonic gas flow rate will increase. The gas flow through a de Laval nozzle is isentropic. In a subsonic flow, the gas is incompressible, and sound will travel through it. At the "throat", where the cross-sectional area is at its minimum, the gas velocity locally becomes sonic, and a condition called the choked flow. As the cross-sectional area of the nozzle increases, the gas begins to expand. The flow accelerates to supersonic speeds where sound waves do not propagate backward through the gas as viewed in the frame of reference of the nozzle with the Mach number greater than 1.0.

The numerical simulation is the process of mathematical modelling, which is designed to predict the behavior or the outcome of a real-world physical system. So, numerical simulation is also called computer simulation. Combined with the finite element or finite volume method, numerical calculation, and image display methods, the purpose of research on engineering or physical problems and even problems in nature can be achieved. Numerical simulation was initially used as a complement to other aspects of research, but when its importance was discovered, it became used quite extensively as a separate subject. Numerical simulations range from running for minutes to hours to days. It can be used to obtain new insights into new technologies or to estimate the performance of systems being too complex for analytical solutions.

In 1888 A.D. Swedish inventor Gustaf de Laval developed the nozzle and used in steam turbines at first [4]. He obtained a British patent for this expanding nozzle in 1889. He then contributed to the study of nozzle shape, design of the nozzle, and strength design. Robert Goddard first used the nozzle as a rocket engine [5]. This gives insight into contemporary rocket research. Most modern rocket engines that employ hot gas combustion use de Laval nozzles. The shape of the nozzle now commonly used in rockets includes bell-shaped or conical. In a high expansion ratio tapering broad nozzle, the high temperature gas produced in the combustion chamber is discharged through an open hole.

This article will explain some aspects of the current research advances on nozzle models in different fields, as well as the numerical simulation methods of the nozzle flow. We will introduce several studies related to nozzle flow and summarize the development and application of nozzle. In Section 2, theoretical research progresses of the nozzle is discussed. The application related advances are detailed in Section 3. Conclusions and prospects are summarized in Section 4.

2. Numerical simulation methods for nozzle flow

2.1. Numerical simulation of nozzle based on Eulerian method

The Eulerian method is also called as Euler representation. It does not study the movement of individual flow particles but is a method to study the fluid movement by studying the flow at a fixed point in space. The Eulerian method regards fluid movement as the change of flow field with time, that is, the time change of spatial velocity distribution. As the application of the Eulerian method, the Euler equation, namely differential equation of motion, is one of the most important basic equations in inviscid fluid dynamics. It refers to the differential equation of motion obtained by applying Newton's second law to inviscid fluid. Euler equation is widely used in the engineering field, especially in fluid
dynamics. In the flow field, the Euler equation can be used for compressible fluids as well as for incompressible fluids - an appropriate equation of state should be used, or the divergence of velocity should be assumed to be zero. For Nozzle flow, supposing the cross section of the nozzle is denoted as \( s(x) \), then the Euler equations are written as:

\[
\frac{\partial (\rho S)}{\partial t} + \frac{\partial (\rho u S)}{\partial x} = 0 \\
\frac{\partial ( \rho u S )}{\partial t} + \frac{\partial ( \rho u^2 + p ) S}{\partial x} = -p \frac{dS}{dx} \\
\frac{\partial ( \rho ES )}{\partial t} + \frac{\partial ( \rho u HS )}{\partial x} = 0
\]  

(1)

Using the primitive variables, Eq. (1) can be transformed as:

\[
\frac{\partial p}{\partial t} + u \frac{\partial p}{\partial x} + \rho \frac{\partial u}{\partial x} = -\frac{\rho u dS}{S} \\
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + \frac{1}{\rho} \frac{\partial p}{\partial x} = 0 \\
\frac{\partial p}{\partial t} + u \frac{\partial p}{\partial x} + \rho \frac{c^2}{2} \frac{\partial u}{\partial x} = -\frac{\rho u c^2 dS}{S} \
\]

(2)

Figure 2. A typical nozzle shape and the definition of cross-section area

2.2. Numerical simulation of nozzle based on Navier-Stokes equations

The Navier-Stokes equations reflect the basic mechanical law of viscous fluid flow, which is of great significance in hydrodynamics. It is a set of nonlinear partial differential equations, which is very difficult and complex to be solved. Before the solution ideas or techniques have not been further developed and breakthrough, it can only be obtained in several special flow problems. Still, in other cases, the approximate solution can be obtained by simplifying the equation. For example, when Reynolds number \( Re \geq 1 \), the viscous force is far less than the inertial force outside the boundary layer of the flow object. The viscous term in the equation can then be ignored; the Navier-Stokes equation is simplified to the Euler equation in ideal flow; in the boundary layer, the Navier-Stokes equation can be simplified to the boundary layer equation, etc. [6]. Due to the advent of the computer, the numerical solution of the Navier-Stokes equation has developed greatly. The concrete equation is as follows:

\[
\frac{\partial \rho}{\partial t} + \text{div}(\rho U) = 0
\]

(3)
\[
\frac{\partial (\rho E)}{\partial t} + \text{div}(\rho EU) = \text{div}(kVT) + \text{div} (\sigma U) \quad (4)
\]

\[
\frac{\partial (\rho E)}{\partial t} + \text{div}(\rho EU) = \text{div}(kVT) + \text{div} (\sigma U) \quad (5)
\]

where equations (3), (4), (5) represent the continuity, momentum, energy equation, respectively. To numerically solve the Navier-Stokes equations using the finite-difference method or finite-volume method, a computation domain, boundary conditions, initial conditions, etc., must be specified. Details can be found in [7].

2.3. Numerical simulation of nozzle based on small disturbance equations

In fluid dynamics, potential flow describes the velocity field as the gradient of a scalar function: the velocity potential. As a result, a potential flow is characterized by an irrotational velocity field, which is a valid approximation for several applications. The irrotationality of a potential flow is due to the curl of the gradient of a scalar always being equal to zero. In the case of an incompressible flow, the velocity potential satisfies Laplace’s equation, and potential theory is applicable. However, potential flows have also been used to describe compressible flows. The potential flow approach occurs in the modeling of both stationary as well as nonstationary flows. Applications of potential flow include: the outer flow field for airfoils [8], water waves [9], electroosmotic flow [10], and groundwater flow [11]. For flows (or parts thereof) with strong vorticity effects, the potential flow approximation is not applicable.

In order to derive the small disturbance equation, a simplification of the full potential equations is done in the case of thin obstacles, such as thin airfoils. We will restrict the discussion to 2D. Since the obstacle is small, it is the effect on the flow. It is insignificant and we consider perturbation to uniform flow with the velocity of magnitude \( U_\infty \) in the \( x \)-direction. The potential has a representation as:

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

The velocities can be recovered from the potential using the following formulas:

\[
\begin{align*}
u &= U_\infty (1 + \Phi_x) \\
v &= U_\infty \Phi_y
\end{align*}
\]

The equation becomes:

\[
\left(1 - M_\infty^2\right) \Phi_{xx} + \Phi_{yy} = 0
\]

which can be simplified into:

\[
\left(1 - M_\infty^2\right) \Phi_{xx} + \Phi_{yy} = 0
\]

The wall boundary condition becomes:

\[
v = \left(U_\infty + u\right) f'(x) = U_\infty f'(x)
\]

where \( f(x) \) is the shape of the airfoil. Z. RUSAK [12] presented some similarity solutions of the small-disturbance equation for a two-dimensional near-sonic potential flow. The analysis is based on the basic similarity solutions of the problem in the hodograph plane. The first-order similarity solution of the velocity potential in the physical plane is described analytically by a parametric representation in terms of the hodograph-similarity variable and function. The second-order similarity term is governed by a linear equation and described analytically as a product of two functions: one is the power of the basic hodograph-similarity function, and the other is a solution of the basic hodograph-hypergeometric equation with a different constant. It is noticed that the application of the present solutions to the far-field approximation of a sonic flow about a thin airfoil results in a relation between Frankl’s special similarity parameter and the hodograph-similarity variable.
3. Nozzle related applications

Kiran utilized both the experiment and computational tools to study the flow fields associated with truncated annular plug nozzles with varying lengths [13]. And he got for all plug lengths considered, the transition of the open base wake to a closed one is observed. The annular plug nozzle flows have also been compared with appropriate linear plug nozzle flows. The three-dimensional relief experienced by annular plug flows results in greater wave interactions on the plug surface as compared with linear plug flows, and in turn, the transition of the base wake is delayed in the case of annular plugs.

Ignacio used numerical simulations of the Euler equations, the analytical solution with the Marble–Candel model, and the quasi-one-dimensional linearized Euler equations model in the frequency domain to study a subsonic nozzle about combustion noise. In the entropy wave generator experiment, the three ways were used to compute the transfer functions of the nozzle and to solve the propagation through the nozzle without the compact nozzle hypothesis and with a strict separation of direct and indirect noise and taking into account the inlet and outlet reflection coefficients. This method gives the right trend for the reference test case and over a wide range of throat Mach numbers for the first time. Using a first-order analysis of the waves generated by the heating device, it has been shown that, for the subsonic cases, direct noise is significant in the experimental setup and that the indirect noise is negligible [14].

Ten-See posed an aeroelastic modelling method to analyse the nozzle transient flow. In that work, computational fluid dynamics have been demonstrated as a powerful analysis and design tool in computing and understanding the underlying transient side load physics. Ten-see found that, when the flow field was not disturbed by the major side load events, two sinusoidal circumferential pressure profiles were computed for the significantly ovalized nozzles. These two sinusoidal curves have a phase shift of 90 degree [15]. It is therefore postulated that the nozzle deformation may not be symmetric in actual simulation. This phenomenon generates circumferential waves that show up as axial pressure oscillations, as shown in the nozzle wall pressure profiles.

Matthew had an important finding in the understanding of the quiet wind-tunnel nozzle with a Mach number equals to 6. This is a study and optimization of a supersonic nozzle model. Matthew chose a ludwieg-tube configuration because it combines very good flow quality with low per-run cost. He then took a driver tube connected to a converging/diverging nozzle whose area ratio sets the Mach number in the test section. The tunnel will be started by a fast-acting shutter valve at the contraction exit instead of the traditional burst diaphragms downstream of the test section [16]. This novel feature leads to several advantages: the starting time will be decreased; the test section can be built to withstand a much lower pressure; the turnaround time between runs will be reduced; there will be no need to tailor the configuration to the stagnation pressure. Two approaches were explored to optimize the length and boundary-layer stability of the diverging portion of the nozzle. All this research is carried out to build a nozzle for the large hypersonic quiet wind tunnel in the University of Notre Dame.

Junhui simulated nozzle boundary-layer separation in highly overexpanded jets. In that study, a method called large-eddy simulations was used. The method is about a model-scale F404 faceted nozzle that was used to operate at a highly overexpanded jet condition. Under this condition, Junhui draws a conclusion about the presence of the turbulence structure, which is an obvious indication that the nozzle boundary-layer has become turbulent, and this demonstrates that the boundary-layer status prior to the separation is very important to the location of the boundary-layer separation. Besides, Junhui investigated the effect of the surface roughness on the boundary-layer separation because it is a factor that cannot be ignored in laboratory experiments, and it is found that this does not affect the location of the boundary-layer separation. It should be mentioned that the nozzle surface configuration or length, the magnitude of the favorable pressure gradient, and other factors should also be taken into consideration to predict the nozzle boundary layer separation [17].

It is reported that natural gas is liquefied by means of the cryogenic effect when natural gas flows through a Laval nozzle at supersonic speed. Wen Yang and Xuewen Cao designed the Laval Nozzle
combined with cubic curve method, real gas equation of state, arc plus line method, and boundary layer viscosity correction. They also studied the flow and liquefaction process of methane in the nozzle and analyzed the effects of inlet temperature, pressure, and back pressure on the liquefaction process of methane. Wen Yang and Xuewen Cao observed that the gas flow in the nozzle reaches supersonic speed and causes low pressure and low temperature, which makes the gas liquefy. The decrease of inlet temperature or the increase of inlet pressure can make the gas liquefy, but the low temperature (below 170k) will make the gas enter the solid phase region. Similarly, when the pressure increases to 2.5MPa, the gas will also enter the solid phase region, which hinders the gas liquefaction process [18]. With the increase of back pressure, the shock wave will enter the nozzle to weaken or destroy the liquefaction process.

The Laval nozzle, which is also widely used in aerodynamics, is adopted in this scheme. A Laval nozzle with inlet diameter $D_1=20\text{mm}$, throat diameter $D_t=15\text{mm}$, and outlet diameter $D_2=19\text{mm}$ is designed in the nozzle of the original cutting gun. By installing a Laval nozzle inside the FG11 cutting gun, Jiaxin Hu found that the gas mixture passing through the nozzle of the cutting gun reaches a supersonic flow of more than twice the Mach number. It is because of this discovery that it is possible to ensure good dynamics of the flame exiting the nozzle of the cutting gun and achieves flame cutting separation at the riser end of large ingots with a diameter of 2 meters. This effectively improves the cutting technology of ingots with large thickness [19].

Research on the numerical simulation of aerodynamic noises of shear flow based on linearized Euler equations. In the numerical simulation, Guobing Fan and Jianming Yan used dispersion-relation-preserving scheme and compact difference scheme of high-order accuracy in space for dispersion. In a difference scheme for the time-related term, the Runge-Kutta method with low-dispersion and low-dissipation was applied to push ahead. Non-reflecting boundary condition was adopted at the far-field boundary [20]. They found that the treatment for dispersion schemes and boundary conditions in this paper could well simulate the propagation process of aerodynamic noises in the shear layer; the shear flow would have an impact on the amplitude and propagation direction of aerodynamic noises in the flow field; for different modes of pipe sound sources, the shear layer would cause different refraction effects; the direction of sound radiation was rather centralized for the single pipe mode and dispersive for the multi-pipe mode.

4. Conclusions and prospects
This article explains the current research advances on the numerical simulation methods for the nozzle flow, as well as the nozzle related applications in different fields. Overall, we could draw the conclusion that nozzle is widely used in the aerospace field, manufacturing field, energy and power field, and so on, which is generally recognized as a critical component of power engines and other energetic generators. Based on the reviews, we can briefly make an outlook on the Laval nozzle and its numerical simulation. In fact, the analysis, numerical simulation, and modelling in different aspects of the Laval nozzle are now more mature. Therefore, researchers have derived practical applications for nozzles at different sound velocities, such as in natural gas liquefaction and ingot cutting. Meanwhile, there have been several enhancements to the methods used to study the interior of the nozzle, such as meshing. From the review on the numerical simulation of nozzle research, we found that most of the existing simulations using one-dimensional, two-dimensional, Euler's method. While for the Navier-Stokes equation, due to it is difficult to be solved hence its application in the nozzle related investigation is limited, so the use of Navier-Stokes equation-based nozzle flow simulation might be a key research direction in the future.

References
[1] Nikhil, D. D., Suyash, S. V., Pratik, R. M., Rutuja, S. J., K., R. J. (2014) THEORETICAL & CFD ANALYSIS OF DE LAVAL NOZZLE, In Proceedings of 4th IRF International Conference. Pune. Pp.61-64
[2] Theodore, S., Henry, M.H. (2007) Steam Turbine Engineering The specialists' series Whittaker's
handbooks for engineers. Whittaker and Company. 1906, Madison.

[3] Zhou, W.X., Huang, J.Q., Zhou, R.Z. (2009) Improvement of Laval nozzle computational model and its verification by whole machine simulation [J]. Journal of Aerodynamics, 24(11):211-216.

[4] Robert, M. N. (1903). The Steam Turbine. Longmans, Green, and Company, London.

[5] Garrett, S. (2000) From Galaxies to Turbines: Science, Technology, and the Parsons Family. Taylor & Francis Group, England.

[6] Zhang, D. L., (2009) Encyclopaedia of China. Encyclopaedia of China Publishing House, Beijing.

[7] Li, K., Chen, H., Zhang, Y., Wang, S.J. (2013) A New Boundary Layer Equation and its Application to the Flow in Turbo-Machinery. http://www.wileyonlinelibrary.com.

[8] Latrou, M., Breitsamter, C., Laschka, B. (2006) Small Disturbance Navier-Stokes Equations: Application of Transonic Two-dimensional Flows Around Airfoils pp.471-478.

[9] Johnson, R.S. (2003) Shallow Water Waves on a Viscous Fluid-The Undular Bore. https://doi.org/10.1063/1.1693764.

[10] Xuan, X.C., Li, D.Q. (2005) Journal of Colloid and Interface Science Volume 289, Issue 1, 1 September 2005, Pages 291-303.

[11] Cao, G.Y. (2010) Power system voltage small-disturbance stability studies based on the power flow equation. ANU Research Publications, pp.873-882.

[12] RUSAK, Z. (1994) Novel similarity solutions of the sonic small-disturbance equation with applications to airfoil transonic aerodynamics. Society for Industrial and Applied Mathematics, Vol. 54, No. 2: pp. 285-308.

[13] Chutkey, K., Vasudevan, B., Balakrishnan, N. (2014) Analysis of Annular Plug Nozzle Flowfield. Journal of Spacecraft and Rockets Vol. 51, No.2. pp.478-490.

[14] Durin, I., Moreaut, S., Poinson, T. (2013) Analytical and Numerical Study of Combustion Noise Through a Subsonic Nozzle. AIAA JOURNAL Vol. 51, No.1. pp.42-52.

[15] Wang, T.S., Zhao, X., Sijun Zhang, S.J., Chen, Y.S. (2014) Development of an Aeroelastic Modeling Capability for Transient Nozzle Flow Analysis. Journal of Propulsion and Power Vol30, No.6. pp.1692-1700.

[16] Lakebrink, T.M., Bowcutt, G.K., Winfree, T. (2018) Optimization of a Mach-6 Quiet Wind-Tunnel Nozzle. Journal of Spacecraft and Rockets Vol.55, No.2. pp.315-321.

[17] Liu, J.H., Khine, Y.Y. (2020) Simulations of Nozzle Boundary-Layer Separation in Highly Overexpanded Jets. AIAA AVIATION Forum. Naval Research Laboratory, Washington D.C. pp.1-23.

[18] Yang, W., Cao, X.W. (2015) Laval Nozzle Design and its Application in Natural Gas Liquefaction. Journal of Xi'an Shiyou University (Natural Science Edition) Vol.30, No.2. pp.1-8.

[19] Hu, J.X. (2016) Application of Laval Nozzle in Large Section Ingot Cutting Technology. Technical Forum of Casting and forging steel division of China First Heavy Machinery Group Corporation, Qiqihar 161040, Heilongjiang Province. pp.257-258.

[20] Fan, G.B., Yan, J.M. (2017) Research on the numerical simulation of aerodynamic noises of shear flow based on linearized Euler equations. © JVE INTERNATIONAL LTD. JOURNAL OF VIBROENGINEERING, 7, VOL. 19, ISSUE 6: 4742-4758