Uluslararası İleri Doğa Bilimleri ve Mühendislik Araştırmaları Dergisi
Sayı 8, S. 266-272, 9, 2024

Araştırma Makalesi

IJANSER

https://as-proceeding.com/index.php/ijanser
ISSN:2980-0811

International Journal of Advanced Natural Sciences and Engineering Researches
Volume 8, pp. 266-272, 9, 2024

Research Article

Application of the Method of Characteristics to Supersonic Nozzle Design and Its Comparison with CFD Analysis

L. Dai[1*] and A. Haddad[1]

[1]*LMANM, Faculty of Science & Technology, Univ. 8 Mai 1945, Guelma-24000, Algeria*

[*]*lyliadai66@gmail.com*

(Received: 13 October 2024, Accepted: 18 October 2024)

(5th International Conference on Innovative Academic Studies ICIAS 2024, 10-11 October 2024)

ATIF/REFERENCE: Dai, L. & Haddad, A. (2024). Application of the Method of Characteristics to Supersonic Nozzle Design and Its Comparison with CFD Analysis, *International Journal of Advanced Natural Sciences and Engineering Researches*, 8(9), 266-272.

Abstract – Supersonic nozzles play a crucial role in the propulsion systems and engines of various vehicles designed for diverse applications. Contoured nozzles, recognized for their efficiency, are the most commonly used profiles today. This study utilizes the Method of Characteristics (MoC) to design the supersonic section of a contoured nozzle profile, employing an isentropic, inviscid, and irrotational supersonic flow model. The objective is to achieve a parallel and uniform flow with a specified exit Mach number. The results are then compared with those from a CFD simulation of the flow field using the Ansys-Fluent platform. By analyzing pressure and Mach number along both the centerline and wall, the findings revealed a good agreement, highlighting specific characteristics of the MoC.

Keywords – Bell-Shaped Nozzle, Method Of Characteristics, Supersonic Flow, Computational Fluid Dynamics, Centerline And Wall Pressure, Centerline And Wall Mach.

I. INTRODUCTION

Supersonic nozzles are vital components in aerospace engineering and propulsion systems, primarily designed to accelerate gases to supersonic speeds. These nozzles are used in applications such as rocket engines, jet engines, and other high-speed propulsion systems, where high-velocity exhaust gases are necessary to generate thrust. A typical supersonic nozzle, commonly known as a de Laval nozzle, consists of three sections: a converging section, which narrows the cross-sectional area and increases subsonic gas velocity; a throat, the narrowest point where the flow reaches sonic velocity (Mach 1); and a diverging section, where the flow expands and accelerates to supersonic speeds, forming what is known as a Convergent-Divergent (C-D) nozzle [1].

The historical development of supersonic propulsion Convergent-Divergent (C-D) nozzles is characterized by major theoretical breakthroughs, pioneering experiments, and practical applications in both military and civilian aerospace engineering. The shape of supersonic nozzles plays a crucial role in determining several key performance metrics [2-3], particularly in how efficiently exhaust gases are accelerated to supersonic speeds, impacting pressure and velocity distributions. The evolution of supersonic nozzle designs has been a gradual process, driven by advancements in aerodynamics, propulsion technology, and aerospace engineering.

Carl Gustaf Patrik de Laval was the first to develop the C-D nozzle, which typically features a converging section that narrows to a throat, followed by a diverging section that expands, increasing the velocity of a steam jet to supersonic speeds [4-5]. These nozzles, known as "de Laval nozzles," have been widely adopted for rocket propulsion. Robert Goddard was a pioneer in combining a de Laval nozzle with a combustion chamber, significantly improving its efficiency and achieving supersonic speeds of up to Mach 7 [6-7]. While rocket propulsion remains the primary application of de Laval nozzles, their use has expanded into other fields as well.

Conical nozzles, featuring a simple conical shape in the diverging section, are among the most basic types of supersonic nozzles. As gases pass through the diverging cone, they accelerate to supersonic speeds. While these nozzles are easy to design and manufacture, they are less efficient at producing uniform flow fields and maximizing thrust compared to more advanced designs. Nevertheless, due to their simplicity, conical nozzles remain widely used in certain applications [8]. In contrast, bell-shaped or contoured nozzles are more efficient than conical nozzles, thanks to their carefully designed curved geometry [9]. The bell shape minimizes losses from shock waves and generates a more uniform exhaust flow. This nozzle design optimizes gas expansion and maximizes thrust, making it ideal for rocket engines where efficiency is crucial [10].

II. SUPERSONIC SECTION NOZZLE DESIGN

The design of the supersonic divergent section is created using the method of characteristics, implemented through a computer program developed in Fortran 77 [11]. The chosen nozzle profile is bell-shaped and modeled using a second-order quadratic curve for the wall. At the throat, the profile consists of two circular arcs with different radii of curvature, which are joined at the throat. The downstream arc is then tangentially connected to the second-order quadratic wall at the attachment point (cf. Figure 1).

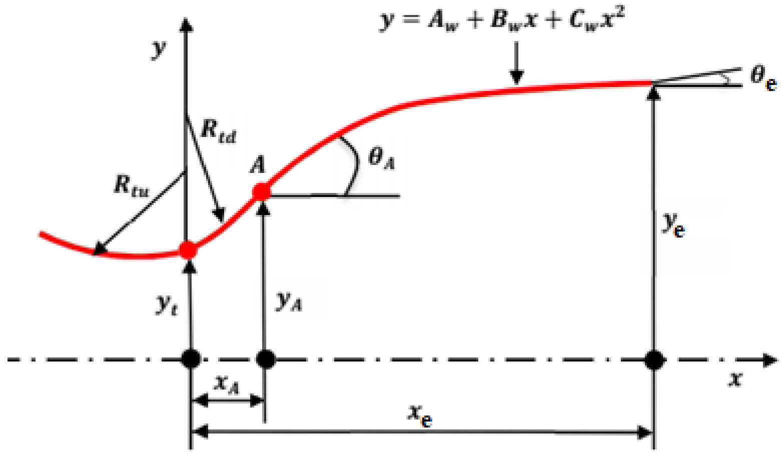

Fig. 1 Bell-shaped nozzle profile contour

The data required to perform the calculations using the Method of Characteristics are presented in Table 1, and are represented by the thermodynamic and geometric data.

Table 1. Thermodynamic and geometrical parameters

Thermodynamic parameters	Geometrical parameters
Atmospheric pressure, P_a=1.013 bars	Throat radius, y_t=0.069 m
Total or stagnation pressure, P_t=69 bars	Upstream and downstream throat radiuses, R_{tu}=0.138m ; R_{td}=0.0345m
Total or stagnation temperature, T_t=2800 K	Attachment angle, θ_A=20°
Specific gas constant, R_G=320 J/kg. K	Exit angle, θ_E=5°
Specific heat ratio, γ=1.2	Exit radius, y_E=0.223m

The results obtained from the computations are detailed in Table 2, providing a comprehensive overview of the output values derived from the Method of Characteristics analysis. These results help to illustrate the geometry of the supersonic nozzle design under the specified conditions.

Table 2. Thermodynamic and geometrical parameters

Geometrical feature	
Attachment point 'A' coordinates	x_A= 0.01180m ; y_A= 0.07108m
Nozzle length	x_E= 0.54 m
2nd-order polynomial coefficients	A_W= 0.06676 ; B_W=0.36882 ; C_W=−0.20535

A unique advantage of the Method of Characteristics lies in its ability to generate the mesh of the domain, and consequently the profile, as the calculations progress. The mesh is formed by the intersections of the right-hand and left-hand characteristic lines. Figure 2 displays these intersections, which serve as the nodes where various solution parameters such as velocities, pressures, temperatures, and densities are computed.

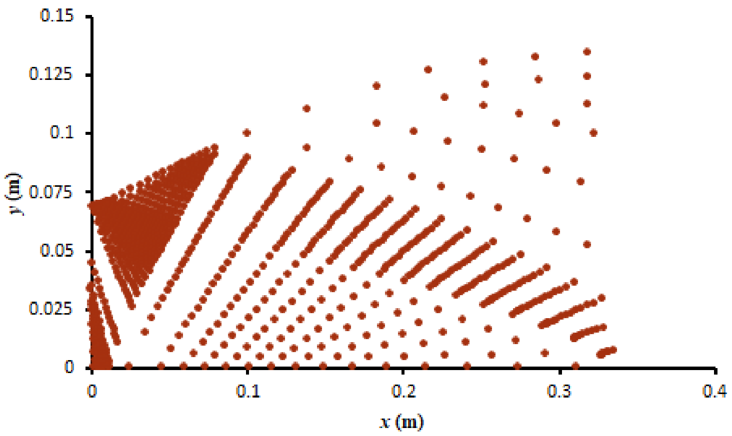

Fig. 2 Diagram of computations illustrating Mach lines

III. SUBSONIC SECTION NOZZLE DESIGN

The main goal of the subsonic convergent section is to accelerate the flow to reach transonic velocities at the throat, which will enable a supersonic expansion in the divergent section. It is attached to the upstream arc of a circle with a curvature radius (R_{tu}), forming the complete profile of the nozzle. The design of the subsonic section employs the Rao technique, which has been developed from various experimental studies and is primarily based on the throat radius [12]. The Rao method generates the subsonic profile as follows:

$$\begin{cases} x = 1.5 \cdot y_t \cdot cos(\theta) \\ y = 1.5 \cdot y_t \cdot sin(\theta) + 2.5 \cdot y_t \end{cases} \quad (1)$$

with:

$$-130° \leq \theta \leq -90° \quad (2)$$

The dimensions and configuration for the converging section, determined through calculations using Rao's method, result in the profile illustrated in Figure 3. This section, when combined with the previously designed divergent supersonic part (Figure 1), finalizes the overall converging-diverging de Laval nozzle, ensuring high-speed flow transitions. The connection between these two sections creates a nozzle that efficiently handles subsonic and supersonic flows in the converging and diverging sections respectively.

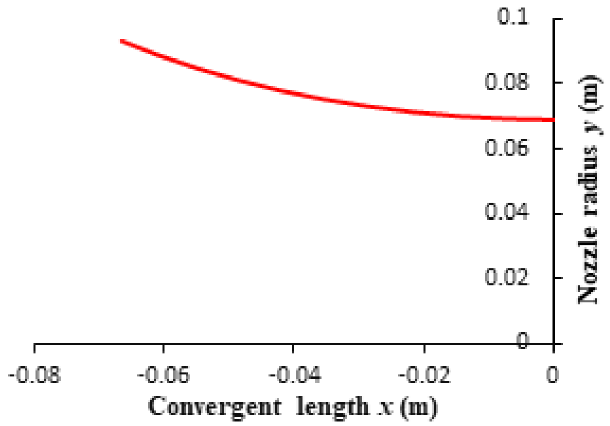

Fig. 3 Profile of the nozzle's convergent section

IV. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

A numerical simulation was conducted to examine the gas flow within the contoured nozzle using the Ansys-Fluent platform [13]. The governing equations for the simulation are based on the fundamental physical principles of conservation of mass, momentum, and energy, which control the behavior of the turbulent viscous flow. The *k-ω* sst turbulence model was integrated into the system to complete the analysis. The computational settings used for the simulation are detailed in Table 3.

Table 3. Nozzle simulation parameters

	Applied features
Model	2D, , stationary, viscous with compressibility effects
Solver	Density based
Turbulence model	*k-ω* sst
Fluid	air, ideal gas
Residual	10^{-6}

The mesh utilized consists of 8,000 structured quadrilateral elements, with 160 elements aligned along the axial direction (x) and 50 elements along the radial direction (y). Mesh refinement was applied near the throat, where sharp gradients in flow properties are anticipated, and along the solid walls to account for the significant viscous effects. Due to the nozzle's symmetrical nature, only half of the fluid domain was simulated. Additionally, a zero transverse velocity condition was imposed along the symmetry plane to preserve the flow symmetry (Figure 4).

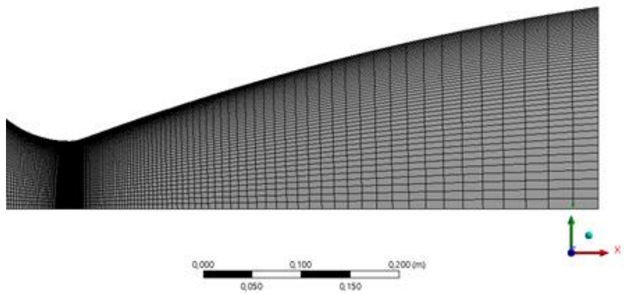

Fig. 4 Quadrilateral structured mesh

V. RESULTS AND DISCUSSION

The solution along the centerline is presented in Figure 5, where the Method of Characteristics (MoC) results are compared with the Computational Fluid Dynamics (CFD) solution obtained using a Finite Volume Method (FVM) approach. The figure highlights a sharp expansion of the flow near the throat, leading to a rapid drop in static pressure. Moving further downstream, the pressure gradually stabilizes. A slight discontinuity is observed, indicating the transition between the downstream arc-of-circle and the nozzle wall. This transition point is labelled as 'A' in Figure 1.

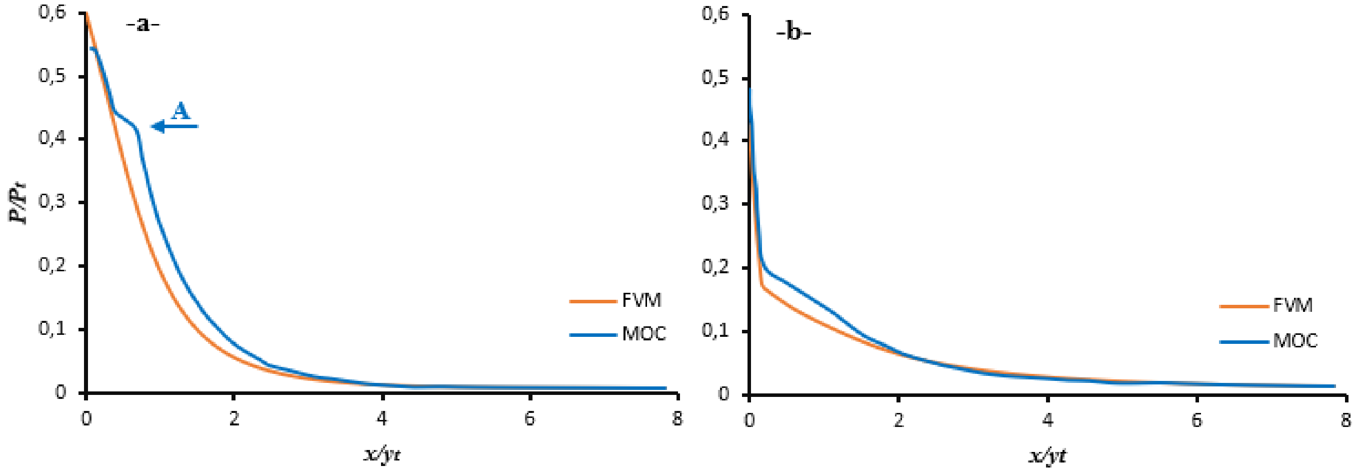

Fig. 5 MoC and CFD solutions in terms of static pressure along (a) centerline and (b) wall

The CFD simulation provides insights into the flow behavior within the nozzle. Figures 6-a and 6-b display the results for static pressure and Mach number respectively. The pressure decreases steadily from the stagnation value of 69 bars at the converging inlet to an average of 0.5 bars at the exit. A more significant pressure drop is observed immediately downstream of the throat, followed by stabilization further along the divergent section, particularly after the attachment point. The flow exits the nozzle nearly axially, aligned with the centerline, maximizing thrust in that direction.

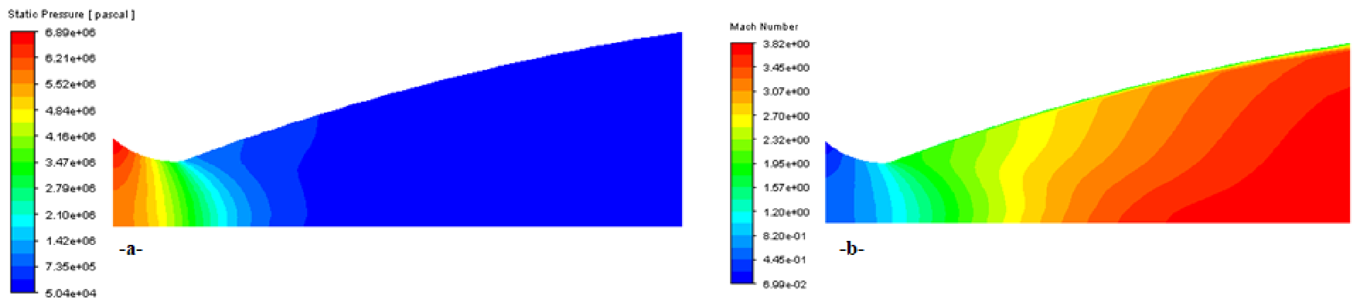

Fig. 6 contour plots of (a) static pressure and (b) Mach number along the nozzle

The velocity vectors representing the flow-field and resulting from its analysis are depicted in Figure 7-a, illustrating the smooth acceleration of flow along the nozzle walls. This indicates satisfactory convergence of the computational method, confirming that the flow expands effectively without the occurrence of shock waves.

Figure 7-b displays the gradual acceleration of the flow at specific cross-sections. It can be observed that a significant portion of the acceleration happens immediately downstream of the throat. Approximately 50% of the flow acceleration occurs within the initial 30% of the nozzle's divergent section. This behavior is a typical characteristic of bell-shaped nozzles. In the remaining part of the divergent section, the flow continues to accelerate and is directed axially to optimize thrust while minimizing the nozzle's length.

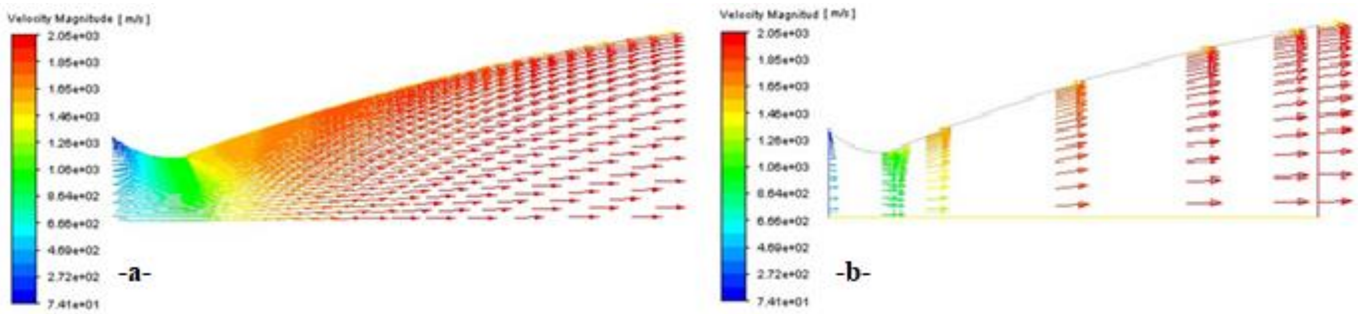

Fig. 7 Velocity distribution along (a) centerline and (b) cross-sections

Table 4 outlines the performance parameters, which are consistent across the results. The mass flow rate remains constant, leading to similar thrust and thrust coefficient values. The 3.22% variation in Mach number is likely due to differences between the Mach number at the exit along the centerline and the wall, which are related to the Method of Characteristics (MoC) computations.

Table 4. Nozzle performance parameters

Performance parameters	**MoC**	**Ansys-Fluent**	**Error (%)**
Developed thrust; [T (N)]	165001.5	166643.1	0.99
Masse flow rate; [$\dot{m}$ (kg/s)]	70.52	70.7	0.25
Thrust coefficient [C_T]	1.60	1.61	0.62
Effective velocity [V_{eff} (m/s)]	2339.78	2357.05	0.73
Exit Mach number [M_e]	3.1	3	3.22
Specific impulse [I_s]	238.5	240.3	0.75

VI. CONCLUSION

Recent developments in propulsion technology are closely linked to significant advancements in nozzle contour design and the aero-thermodynamic analysis of propulsion nozzles, especially in their supersonic sections. Achieving maximum exit Mach numbers and generating high thrust while maintaining compact and lightweight designs require the optimization of all engine components, with the nozzle playing a critical role in producing thrust. This research focuses on comparing two distinct methods for analyzing the supersonic flow field within a bell-shaped supersonic propulsion nozzle. The Method of Characteristics (MoC) and Finite Volume Method (FVM) approaches produced comparable results regarding pressure and Mach number distributions. The comparison of performance parameters also showed similar outcomes. One key advantage of the MoC method is its ability to design a supersonic section profile based on a desired exit Mach number.

REFERENCES

[1] 1. Khare, S., Saha, U. K. Rocket nozzles: 75 years of research and development, Sādhanā, 46(2), 76 (2021). https://doi.org/10.1007/s12046-021-01584-6

[2] Quintao, K. K., Design optimization of nozzle shapes for maximum uniformity of exit flow, FIU electronic theses and dissertations, Paper 779, (2012). https://doi.org/10.25148/etd.FI12120505

[3] Turner, J. L., Rocket and spacecraft propulsion: principles, practice and new developments, Springer Science & Business Media, (2008). ISBN 978-3-69202-7

[4] Anderson, J. D., Modern compressible flow: with historical perspective, 4th ed., McGraw-Hill Education, 2003. ISBN 978-1-260-57082-3

[5] Sutton, G. P., Biblarz, O., Rocket propulsion elements, John Wiley & Sons, 2011. ISBN 978-0-470-08024-5

[6] Goddard, R.H., Rockets, Courier Corporation, 2022. ISBN 0-486-42537-1 (pbk)

[7] Jayaprakash, P., Dhinarakaran, D., Das, D., Design and analysis of a rocket C-D nozzle, International Journal of Health Sciences, 6(S5), pp. 3545–3559, (2022). https://doi.org/10.53730/ijhs.v6nS5.9404

[8] Gayathri, N., Senthilkumar, P., Dineshkumar, K., Purusothaman, M., Sriharan, B. CFD analysis on flow through nozzle of conical type and truncated conical plug type, Materials Today: Proceedings, 72, 2387-2394 (2023). https://doi.org/10.1016/j.matpr.2022.09.407

[9] 3. Arrington, L., Reed, B., Rivera, Jr, A. A performance comparison of two small rocket nozzles. In 32nd Joint Propulsion Conference and Exhibit. (1996). https://doi.org/10.2514/6.1996-2582
[10] Prjadko, N. S., Strelnikov, G. O., Ternova, K. B. Research of supersonic flow in shortened nozzles of rocket engines with a bell-shaped tip, Space Science and Technology, 30(1), 3-13. (2024). https://doi.org/10.15407/knit2024.01.003
[11] 5. Haddad, A., Moss, J. B. Aerodynamic design for supersonic nozzles of arbitrary cross section, Journal of Propulsion and Power, 6(6), 740-746 (1990). https://doi.org/10.2514/3.23280
[12] Uyeki, D. Y. K., A design method for a supersonic axisymmetric nozzle for use in wind tunnel facilities, MSc in Aerospace Eng, Dept of Aerospace Eng, Dan José University, 201, (2018)
[13] 6. ANSYS, Inc., Fluent, Release 21.2, ANSYS, Inc., 2022.