The impact of the Laval nozzle shape on thrust production, using the method of characteristics

Andrei - Ionut PARASCHIV*

*Corresponding author
Department of Propulsion & Power, Faculty of Aerospace Engineering,
Technische Universiteit Delft,
Kluyverweg 1, 2629 HS, Delft, Netherlands,
A.Paraschiv@student.tudelft.nl

DOI: 10.13111/2066-8201.2024.16.1.8

Received: 05 December 2023/ Accepted: 23 January 2024/ Published: March 2024

Abstract: *This investigation examines the influence of Laval nozzle geometry on thrust production using the method of characteristics. It explores various divergent section configurations, analyzing their impact on nozzle design and performance. Comparative analysis of software tools, study of supersonic flow phenomena, and application of the method of characteristics form the core of this research. Findings showcase strong agreement between computational tools, emphasizing the Mach number's role in divergent section shape variations and thrust force. Additionally, the study scrutinizes over-expansion and under-expansion phenomena, validated through computational simulations. Future research should be aimed at enhancing computational methodologies and investigating additional parameters affecting nozzle performance, promising advancements in rocket propulsion technology.*

Key Words: *Method of characteristics (MoC), Computation Fluid Dynamics (CFD), Converging-diverging nozzle, Bell nozzle, Nozzle shape, Supersonic flows, Shockwaves, Thrust*

1. INTRODUCTION

Rocket motors, vital to space exploration, owe a significant part of their thrust generation to the Laval nozzle. Nozzles are critical components in rocket systems, serving as an exit point for high-speed exhaust gases. They play a pivotal role in transforming thermal energy into kinetic energy, propelling the rocket forward. Efficient nozzles contribute directly to the rocket thrust, fuel efficiency, and overall mission success.

Among its components, the divergent section holds pivotal importance as it facilitates the acceleration of fluids from a sonic state within the critical section to a supersonic regime at the nozzle exit. This transformation plays a defining role in the efficiency and performance of the propulsion system.

The accurate design and optimization of this crucial divergent section stand as a focal point in rocket propulsion. To achieve optimal velocity and thrust, the Laval nozzle must be meticulously designed. In pursuit of this objective, the method of characteristics proves to be a powerful tool. As stated in [1], this method, which is based on minimal input parameters, has the ability to generate the entire geometry of a Laval nozzle divergent section.

This study embarks on a comprehensive exploration of diverse configurations within the divergent section of Laval nozzles, employing the method of characteristics. Starting from the thermodynamic and geometric parameters within the critical section of the nozzle and the desired Mach number at the nozzle exit, this research aims to scrutinize various aspects related to nozzle design and performance.

Throughout this work, several key subjects will be examined: comprehensive exploration of the characteristic method, its mathematical model, and its application in computational algorithms; Investigation of shock and expansion waves, which play a crucial role in analyzing the flow within over-expanded or under-expanded nozzles; Comparative analysis of different software tools used for generating Laval nozzle geometries; Simulation of flow through a Laval nozzle using ANSYS Fluent.

Reflecting on the present status of Laval nozzles, as seen in papers such as [2] and [3], it's evident that various accepted configurations exist, typically featuring a convergent zone, a critical section with a minimum diameter, and a divergent zone. They differentiate themselves through the shape of the converging section. This way, three main options are found: conical, bell, and Aerospike nozzle.

Building upon prior works by Kumar and Ogalapur [4] and Fernandes, Souza, and Afonso [5], this study aims to lay the groundwork for an in-depth exploration into Laval nozzle intricacies, computational methodologies, and critical factors governing efficiency and performance. Through meticulous analysis of divergent section configurations and computational techniques, this research endeavors to make substantial contributions to the advancement of rocket propulsion technology.

2. SUPERSONIC FLOW ANALYSIS

The convergent-divergent nozzle, also known as the Laval nozzle after Gustaf Patrik de Laval, a Swedish engineer who implemented this configuration in a high-speed steam turbine, is utilized to generate supersonic flow. The conservation laws are applied in two different sections of the nozzle to study the flow in any type of nozzle. The continuity equation and energy equation are processed:

$$\dot{m} = \rho A V = const. \tag{1}$$

$$h_0 = h + \frac{V^2}{2} = const. \tag{2}$$

where $\dot{m}$ is the mass flow rate through the nozzle, A is the area of a nozzle section, V is the flow speed through that section, and h_0 and h are the total and static enthalpy, respectively.

By differentiating these equations and introducing the first law of thermodynamics for adiabatic flow, relationships describing pressure and velocity variations are derived. These relationships are utilized to determine the nozzle sections, showcasing the interdependence of area, velocity, and pressure in various nozzle designs. The first important result is the following equation:

$$\frac{dA}{A} = (M^2 - 1)\frac{dV}{V} \tag{3}$$

where M is the flow's Mach number in a given section A.

Equation (3) represents an interdependence between flow speed, Mach number and section area, which is the basis for designing various types of nozzles used these days. The equation helps to differentiate between three flow regimes:

- In the subsonic regime ($M < 1$), the fluid particles will accelerate if the nozzle narrows and will slow down if the section area increases.

- In the transonic flow regime ($M \approx 1$), the section will be at its minimum.
- In the supersonic regime ($M > 1$), the section area and the fluid velocity are directly proportional.

The other important equations derived from the conservation equations are those correlating the thermodynamic properties between two different sections of the nozzles: As stated above, these are derived using the first law of thermodynamics and assuming isentropic flow through the nozzle.

Several flow configurations can be analyzed for a Laval nozzle. Their pressure distribution is seen in Figure 1, while each particular case in analyzed afterwards.

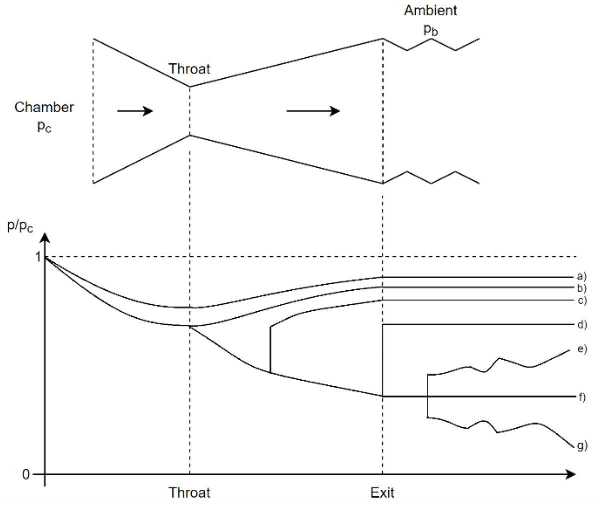

Figure 1. Pressure distribution throughout a Laval nozzle

Firstly, case **(a)** refers to the situation where the fluid enters the nozzle at a subsonic speed but does not reach $M = 1$ at the minimum area section. Consequently, it will decelerate within the divergent section. By decreasing the ambient pressure (p_b), the velocity increases throughout the nozzle [6]. Continuing the decrease in ambient pressure, case **(b)** happens, where the velocity within the nozzle will increase until it reaches $M = 1$ at the minimum area section. Consequently, the flow becomes choked, and no matter how much the ambient pressure decreases, the flow remains in a transonic regime at the minimum section [6].

If the decrease in ambient pressure continues, the pressure in the divergent area of the nozzle will also decrease, allowing the velocity to increase after the critical zone, referred to in case **(c)**. This enables the formation of a supersonic flow that generates a normal shockwave. This will abruptly reduce the speed back to a subsonic regime. In this scenario, by decreasing or increasing the ambient pressure, the supersonic flow region in the divergent section of the nozzle can be expanded or reduced [6]. Continuing the decrease in ambient pressure can extend the supersonic flow region all the way to the nozzle exit, represented by case **(d)**. Consequently, the shockwave will only form at the end, and the velocity will be much higher. However, after the shockwave, the flow regime will still be subsonic [6].

Decreasing the ambient pressure even further causes the shockwave to form outside the nozzle, creating a series of shockwaves reflected in the airflow. In this scenario, there will be an alternation between the supersonic and subsonic regimes. This type of flow is also known as over-expanded flow because the fluid exit pressure is lower than the pressure of the ambient ($p_e < p_b$). This can be observed in Figure 2 [6].

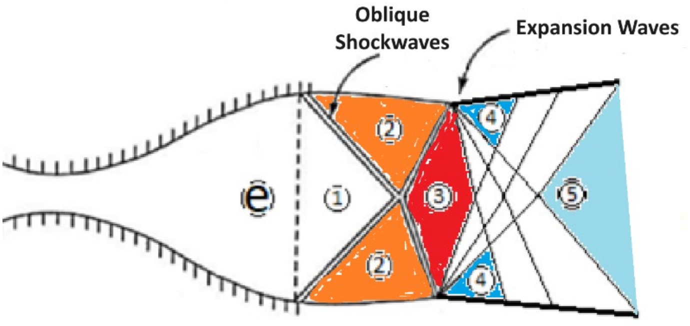

Figure 2. Over-expanded nozzle flow configuration – case **(e)**

Continuing the pressure decrease will lead to the point where the exit pressure from the nozzle equals the ambient pressure (perfect expansion). Consequently, the shockwaves vanish, and the flow becomes uniform. This scenario is ideal and serves as a design condition for any nozzle application [6].

If the pressure decreases further beyond this point, there will again be a disparity between the exit pressure and the ambient pressure ($p_e > p_b$). This case is also known as under-expanded supersonic flow. Consequently, due to pressure differences, expansion waves will occur, accelerating and redirecting the flow outward, forming a kind of plume. This creates another set of complex shockwaves and expansion fans, as represented in Figure 3 [6].

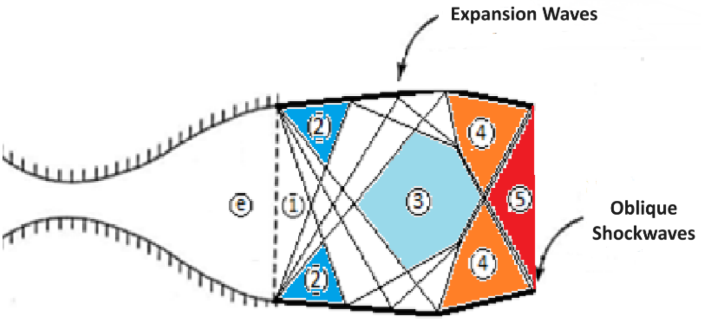

Figure 3. Under-expanded nozzle flow configuration – case **(g)**

Both cases of over-expansion and under-expansion cause thrust losses due to the flow instabilities created by the creation of shockwaves. Therefore, of all of the aforementioned cases, the most sought-after scenario is case **(f)** – the design condition.

That being said, a MATLAB code is created with the aim of determining the parameters of the shockwaves created by an over-expanded or under-expanded nozzle. Its results will be compared to a Java application created by NASA, which can be found on [7].

3. METHOD OF CHARACTERISTICS

The method of characteristics is a natural approach for solving hyperbolic equations. It involves tracing discontinuities within a computational domain along characteristic directions. This method simplifies partial differential equations along characteristic directions into ordinary differential equations, requiring the solution of a linear equation system for each point in the computational grid. It is based on ideal conditions like inviscid, steady, and supersonic flow. Practically, characteristic lines are treated as curves. Calculating these curves can be challenging, so, for computational simplicity, they are often approximated as straight segments [8]. The term *inviscid* implies the absence of viscous effects in fluid dynamics. In the context of nozzles, considering inviscid flow helps simplify mathematical models and computational analyses. This simplification is crucial for predicting and optimizing the complex flow patterns in rocket engines. The mathematical model starts from the simplified Navier-Stokes to which the divergence equation is applied, resulting in:

$$\frac{\partial u}{\partial x}\left(1-\frac{u^2}{a^2}\right)+\frac{\partial v}{\partial y}\left(1-\frac{v^2}{a^2}\right)-\frac{uv}{a^2}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)+\varepsilon\frac{v}{y}=0 \tag{4}$$

where $\varepsilon = 1$ when designing an axis-symmetric nozzle and $\varepsilon = 0$ when designing a planar nozzle (purely theoretical).

By using eigenvectors and eigenvalues theory, the slope of the characteristic lines can be derived, resulting in:

$$\lambda_{\pm} = tan(\theta \pm \mu) \tag{5}$$

where the subscript stands for the nature of the characteristic line (positive or negative), θ is the flow turn angle, and μ is the Mach angle, which is defined as $\mu = arcsin\left(\frac{1}{M}\right)$.

The other important relation will be derived using algebraic manipulation and Crocco's theorem. This is called the characteristic equation and looks like this:

$$d\theta = \mp\frac{dV}{V}cot\,\mu \pm \varepsilon\frac{dx}{y}\frac{sin\,\theta\,sin\,\mu}{cos(\theta\mp\mu)}\mp\frac{ds}{2kR}sin\,2\,\mu \tag{6}$$

where the second term of the right-hand side makes the difference between the axis-symmetric case and the planar case, and the third term stands for the rotational nature of the flow. Therefore, by assuming irrotational flow, equation (6) becomes:

$$d\theta = \mp\frac{dV}{V}cot\,\mu \pm \varepsilon\frac{dx}{y}\frac{sin\,\theta\,sin\,\mu}{cos(\theta\mp\mu)} \tag{7}$$

Furthermore, the equation can be simplified more based on the type of analysis needed: axis-symmetric or planar. As stated before, $\varepsilon = 1$ when designing an axis-symmetric nozzle. Therefore, equation (7) turns into:

$$d\theta = \mp\frac{dV}{V}cot\,\mu \pm \frac{dx}{y}\frac{sin\,\theta\,sin\,\mu}{cos(\theta\mp\mu)} \tag{8}$$

On the other hand, designing a planar (theoretical) nozzle assumes $\varepsilon = 0$. With this being said, equation (7) becomes:

$$d\theta = \mp\frac{dV}{V}cot\,\mu \tag{9}$$

Using the relation between Mach angle and Mach number, and, then integrating, equation (9) turns into the compatibility equation for any given characteristic line:

$$K_{\pm} = \theta \mp \nu(M) = \text{constant} \tag{10}$$

where $\nu(M)$ is the Prandtl-Meyer angle and is defined as:

$$\nu(M) = \sqrt{\frac{k+1}{k-1}}arctan\sqrt{\frac{k-1}{k+1}(M^2-1)} - arctan\sqrt{M^2-1} \tag{11}$$

where k is the specific heat ratio of the fluid.

With the derivation complete, the focus shifts to programming. One more MATLAB code is created based on the equations for determining the geometry of the divergent section of the nozzle in the planar case. All the results will be, in turn, compared to the same Java application used for the study of shockwaves [7].

4. RESULTS

In this section, different shapes of the nozzle will be compared with respect to the thrust they can produce. This will be done based on the thrust equation of a rocket:

$$F = \dot{m}V_e + (p_e - p_H)A_e \tag{12}$$

where $\dot{m}$ is the mass flow rate through the nozzle, the subscript e stands for parameters at the exit section of the nozzle, and p_H refers to the pressure of the ambient at altitude H [9].

Using the same input parameters for both MATLAB and NASA codes, two nozzle shapes are generated. The inputs are: total pressure and temperature in the combustion chamber ($p_{0,c} = 3.447bar$, $T_{0,c} = 555.555K$), the altitude ($H = 10668m$), the design exit Mach number ($M_e = 2$), the number of characteristic lines used ($n = 25$), and the geometric parameters of the nozzle throat ($h_{th} = 0.254\ m$). The resulting configuration, from the NASA code, can be seen in Figure 4, also with the characteristic lines drawn, while the comparison between the configurations of the two codes used can be observed in Figure 5.

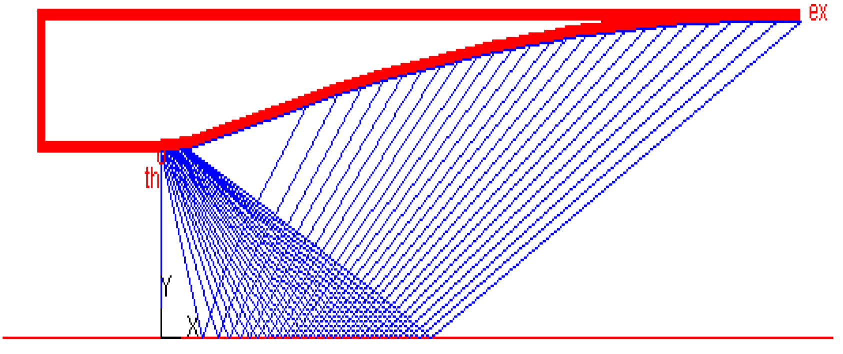

Figure 4. Shape of the nozzle, generated by NASA's code

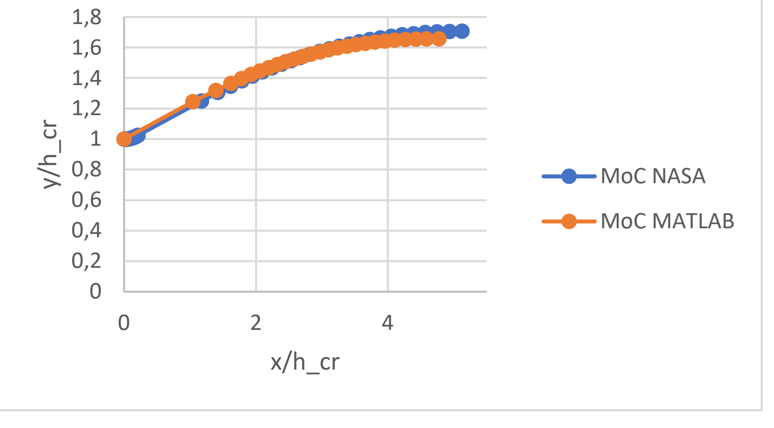

Figure 5. Shapes produce by both codes (adimensionalized w.r.t. throat geometry)

Based on this first geometry, an analysis of nozzle shape variation was conducted. The aim was to find out how the different inputs would affect the final shape generated by the code. Firstly, the design exit Mach number was varied, while keeping all other inputs constant, and the results can be seen in Figure 6.

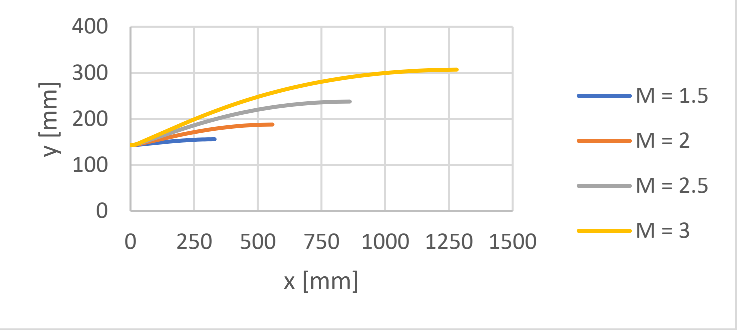

Figure 6. Different nozzle shapes for different exit Mach numbers

In Figure 6 it can be seen, that the area of the exit section of the nozzles increases with the exit Mach number, which is to be expected, because the gases need to undergo a longer expansion process to reach higher speeds. Consequently, looking at equation(12), it is also expected that the thrust would be higher for a higher speed. Next, in Figure 7, the thrust variation with the exit Mach number can be observed.

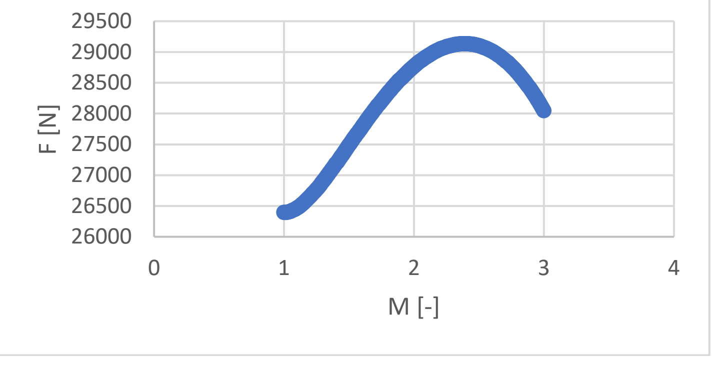

Figure 7. Thrust variation with exit Mach number

It can be seen that, for the inputs provided, the maximum thrust is actually obtained for $M_e \cong 2.39$. The drop in thrust after this value is caused by the over-expansion phenomenon, described by shockwaves created by the difference in pressures. For the next part, the over-expansion and under-expansion phenomena will be studied. Both cases will be analyzed at the same altitude ($H = 10668m$, $p_H = 0.2384bar$). For this, it is already known that perfect expansion happens for $M_e \cong 2.39$. Therefore, in order to study the over-expansion phenomenon, a higher Mach number is required. This has been chosen to be $M_e = 3$. For this value, the static exit parameters are $p_e = 0.091bar$ and $T = 207K$. The shockwave configuration is presented in Figure 8, generated by NASA's code.

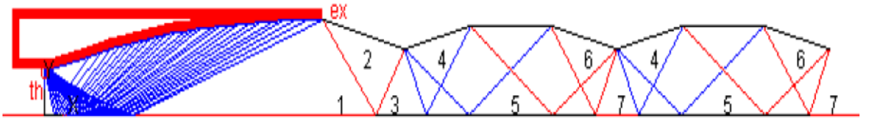

Figure 8. Shockwave configuration for the over-expansion phenomenon

The values obtained based on this configuration are presented and compared with the MATLAB code in Table 1.

Table 1. Shockwave parameters' values (over-expansion)

Zone	Pressure [bar]		Temperature [K]		Mach number [-]	
	NASA	*MATLAB*	*NASA*	*MATLAB*	*NASA*	*MATLAB*
1	0.091	0.091	207	207	3	3
2	0.239	0.2384	276	275.4187	2.327	2.3292
3	0.529	0.5283	347	346.4414	1.789	1.791
4	0.239	0.2384	280	279.3390	2.295	2.2964
5	0.092	0.0817	216	209.0689	2.899	2.9762
6	0.239	0.2384	280	289.018	2.295	2.2172
7	0.529	0.5667	357	372.0243	1.789	1.6175

For the under-expansion case, the analysis is done for $M_e = 2$, which corresponds to pressure $p_e = 0.44bar$ and temperature $T = 317K$. The shockwave configuration is presented in Figure 9, generated by NASA's code.

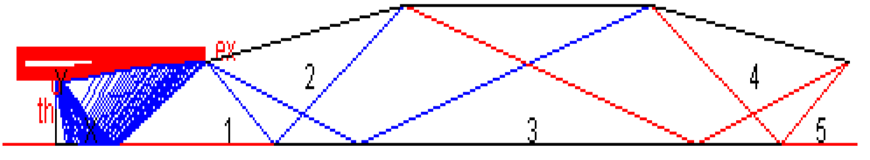

Figure 9. Shockwave configuration for the under-expansion phenomenon

The values obtained based on this configuration are presented and compared with the MATLAB code in Table 2.

Table 2. Shockwave parameters' values (under-expansion)

Zona	Pressure [bar]		Temperature [K]		Mach number [-]	
	NASA	*MATLAB*	*NASA*	*MATLAB*	*NASA*	*MATLAB*
1	0.44	0.44	317	317	2	2
2	0.239	0.2384	269	268.5682	2.386	2.3861
3	0.118	0.1059	222	215.629	2.83	2.9008
4	0.239	0.2384	269	272.6219	2.386	2.3513
5	0.44	0.4756	317	331.6887	1.99	1.8932

For further verification of the results, a CFD simulation was performed in ANSYS Fluent, the geometry used was the one created using $M_e = 2$, at the ambient pressure of $p_H = 0.2384 bar$) in order to test the under-expanded nozzle. The results of the simulation are presented in figures Figure 10 through Figure 13.

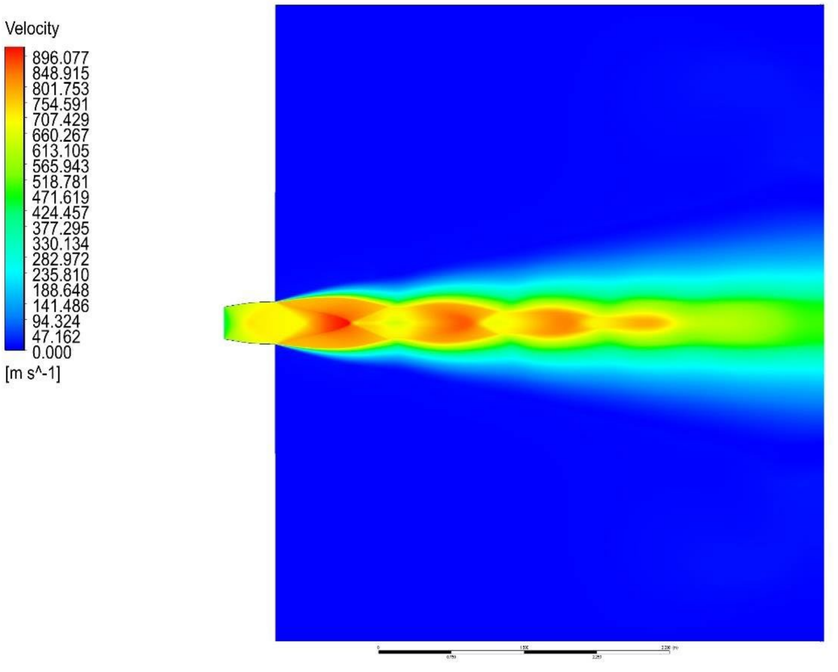

Figure 10. Velocity distribution

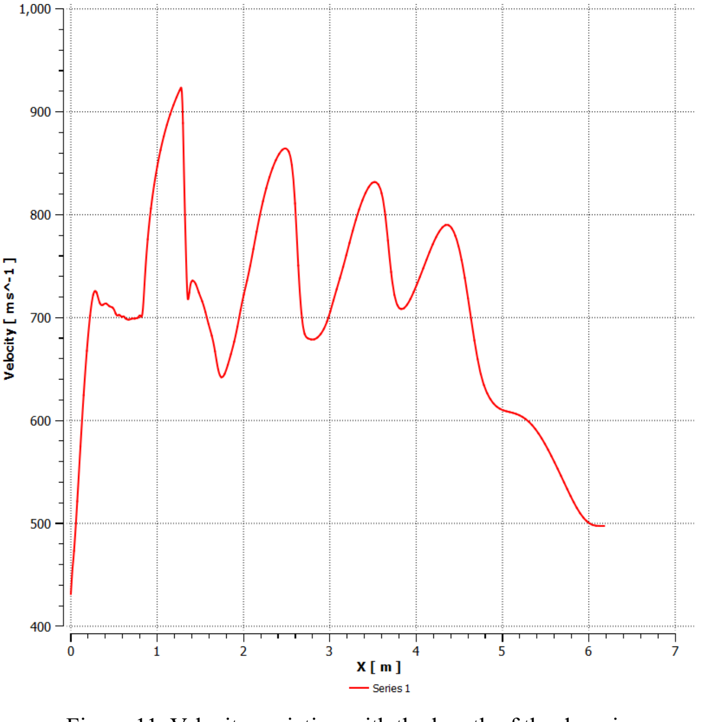

Figure 11. Velocity variation with the length of the domain

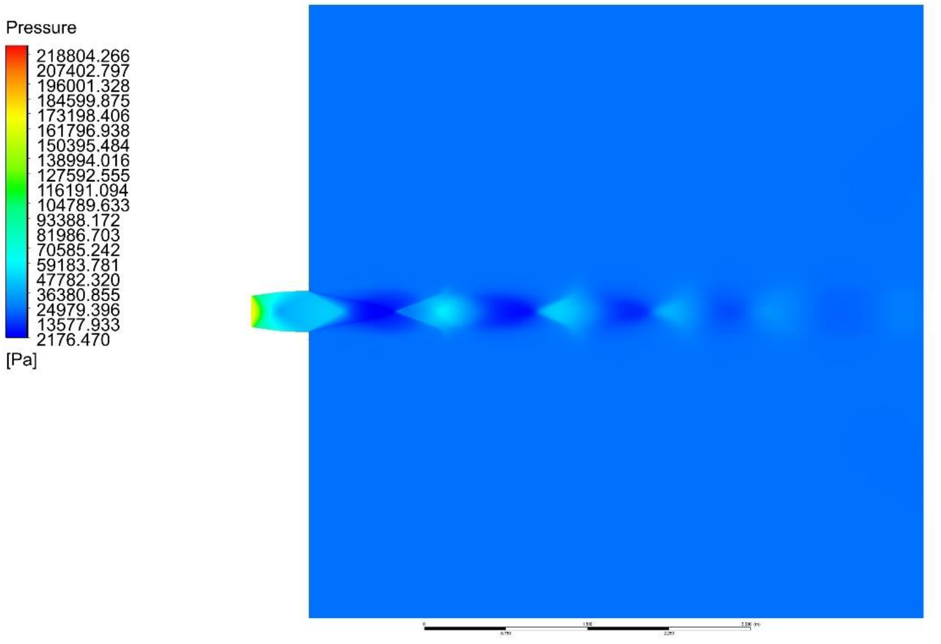

Figure 12. Pressure distribution

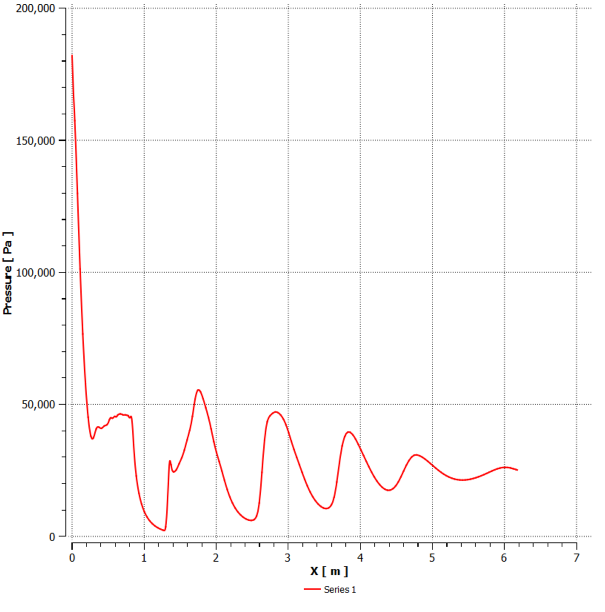

Figure 13. Pressure variation with the length of the domain

5. CONCLUSIONS AND DISCUSSIONS

In this work, the method of characteristics was used to create geometries for a Laval nozzle, with the aim of finding the optimal shape for a given case. The novelty introduced in this study, compared to similar ones, involved utilizing multiple programs across different programming languages (MATLAB and Java) to compare both geometric and thermodynamic/cinematic parameters for various resultant Laval nozzle configurations.

For the planar case of the characteristic method (the focus of this study), a comparison between the MATLAB program and NASA’s application was created, revealing relative percentage errors of less than 7%. When comparing the nozzle shapes created by both programs, it can be seen, in Figure 5, that they are very similar.

In studying configurations resulting from running the two programs with various initial data, certain parameters were found to play a more significant role in geometry variation. For instance, the Mach number influenced both the divergent section length and the exit section radius, as well as the thrust force generated by the nozzle. The maximum thrust force was observed when the nozzle exit pressure equaled the ambient pressure. By varying the Mach number, the optimum value was determined, resulting in a complete expansion within the nozzle and the generation of the corresponding divergent section geometry using the method of characteristics. Consequently, an optimal configuration of a Laval nozzle generating maximum thrust at an altitude of $H = 10668\,m$ was identified.

This study also highlighted the possibility of over-expanded or under-expanded flows occurring based on whether the nozzle exit pressure was higher or lower than the ambient pressure, as specified in [10]. These flow types were analyzed in detail using the MATLAB program developed and NASA's application, with the thermodynamic and kinematic parameters of different flow zones being compared. The results showed a relative percentage error of less than 10%.

Further analysis involved simulating the flow using the ANSYS Fluent software for a supersonic jet generated by an under-expanded nozzle created earlier. The comparison of results and graphs between this simulation and calculations from both programs indicated significant similarities in the behavior of the supersonic jet.

Future research avenues for advancing Laval nozzle optimization encompass fluid composition effects, surface characteristics' influence, thermal impacts on design, and computational methodology enhancements. Exploring how varying fluid compositions and surface conditions affect nozzle efficiency could offer key insights. Additionally, investigating temperature effects and employing advanced computational fluid dynamics methods could refine accuracy. Multi-objective optimization approaches and unconventional geometries also hold promise. Validating theoretical findings through practical applications and environmental impact assessments remains crucial for real-world implementation and sustainability goals. These avenues collectively promise significant strides in nozzle optimization, impacting aerospace and related industries.

The adaptability of inviscid nozzles in varying atmospheric pressures presents an advantage, particularly for aerospike engines experiencing dynamic changes during flight. As stated in [11], designing these nozzles for aerospike engines presents challenges and opportunities, demanding a delicate balance of geometric precision, computational modeling, and a profound understanding of fluid dynamics to ensure optimal thrust across diverse flight conditions.

Looking ahead, exploring the application of inviscid nozzles within aerospike rocket engines stands as a cutting-edge approach to address atmospheric fluctuations during flight. By integrating innovation with fundamental fluid dynamics principles, engineers aim to unlock heightened performance, efficiency, and adaptability, pushing the boundaries of space exploration into new frontiers.

REFERENCES

[1] E. M. Landsbaum, Contour Nozzles, *ARS Journal*, **30**(3), https://doi.org/10.2514/8.5045, 1960

[2] R. D. Rugescu, C. Coman, A. Bogoi, I. Farcasan & D. Constantinescu, Hybrid Rocket Engine Novel Technology for the Orbital Launcher NERVA, *3rd International Conference on Experiments/Process/System Modeling/Simulation & Optimization*, *1*, 249–255, 2009.

[3] G. V. R. Rao, Recent Developments in Rocket Nozzle Configurations, *ARS Journal*, **31**(11). https://doi.org/10.2514/8.5837, 1961

[4] A. Kumar & S. G. Ogalapur, Design of Minimum length nozzle by Method of Characteristics, *Anuj Kumar*, **8**, 2020.

[5] T. Fernandes, A. Souza & Afonso, A shape design optimization methodology based on the method of characteristics for rocket nozzles, *CEAS Space Journal*, **15**(6), https://doi.org/10.1007/s12567-023-00511-1, 2023.

[6] W. J. Devenport, *Nozzle Applet*, https://Www.Engapplets.vt.Edu/Fluids/CDnozzle/Cdinfo.Html, 2001, January 3.

[7] T. Benson, *Undergraduate Computer Programs*, https://Www.Grc.Nasa.Gov/Www/k-12/UndergradProgs/Index.Htm, 2022, April 11.

[8] A. Bogoi, S. Dănăilă & D. Isvoranu, *Ecuațiile Generale de Transport ale Dinamicii Fluidelor*, Monitorul Oficial, 2021.

[9] G. P. Sutton & H. S. Seifert, Rocket Propulsion Elements, *Physics Today*, **3**(2), https://doi.org/10.1063/1.3066790, 1950

[10] A. Bogoi, R. D. Rugescu, V. I. Misirliu, F. R. Bacaran & M. Predoiu, Inviscid Nozzle for Aerospike Rocket Engine Application, *Applied Mechanics and Materials*, **811**, https://doi.org/10.4028/www.scientific.net/amm.811.152, 2015

[11] M. Masdari, M. Tahani & F. Mohammadi Fard, Designing an axisymmetric aerospike nozzle based on muddied MOC, *Scientia Iranica*, **26**(2), https://doi.org/10.24200/sci.2018.5065.1072, 2019.