Development of a Laval nozzle for a cold gas propulsion system

To cite this article: O Dumitrescu et al 2018 IOP Conf. Ser.: Mater. Sci. Eng. 400 042016

View the article online for updates and enhancements.

You may also like
- Combining methods for end-to-end calculation of gas flow in a Laval nozzle
  R E Dushkov and R I Dzerzhinsky
- Improvement of the smelting efficiency based on the use of composite nozzles in the oxygen lances of high-capacity converters
  V V Solonenko, E V Protopopov, S V Feiler et al.
- Numerical study on subsonic-supersonic Laval nozzle
  Xingyue Ji, Junshen Zhi and Hongyu Pan
Development of a Laval nozzle for a cold gas propulsion system

O Dumitrescu, B Gherman and T Tipa
National Research and Development Institute for Gas Turbines COMOTI, 220 D Iuliu Maniu Bvd., Bucharest, Romania
E-mail: oana.dumitrescu@comoti.ro

Abstract. This paper presents a study regarding the calculation and design of a Laval nozzle, and its influence on the performances obtained. For this research, numerical simulation were conducted using the commercial software ANSYS CFX to verify the geometry and to compare it with the analytical results. This study is part of a research project that develops an advanced solar thermal propulsion system; the main purpose is to increase the operation life of a satellite using focused solar light. Two geometries are analyzed in this paper: one with a cone shape and the other generated using the characteristics method. The working fluid is nitrogen gas, which was used due to its inert properties and high molar mass. For the numerical simulations an exterior domain has been taking into account, to capture the jet and to observe how is evolving into the atmosphere. Several cases were studied; temperature was the main parameter varied from one case to another. As a result of the numerical simulations, it was found that the nozzle shape and temperature at the inlet to the convergent section influences performance of the propulsion system in a significant manner. Mach diamonds form because of the supersonic exhaust of the nozzle, and can be seen how their intensity is decreasing in magnitude near the domain outlet.

1. Introduction
One of the most important components of the rocket is the nozzle, whose efficiency affects the rocket overall thrust and efficiency as well as stability of the turbopump and/or combustion chamber. On an ideal design conditions, the exit pressure must match the ambient one, so a shear layer must exist between the high-speed nozzle flow and the ambient (adapted nozzle) [1]. This will lead to the maximum thrust possible obtainable by the nozzle. Increasing the expansion ratio, leads to an increase of thrust [2] but at the cost of added weight. Other aspects such as optimization or minimizing the losses in the nozzle, the thrust of a rocket can be increased [3].

The shape of the nozzle influences the way the expansion of the exhaust gas is transformed into linear motion. The simplest form, conical nozzle, has half an angle of about 15°, which leads to an efficiency of 98% [4]. Smaller angles provide a slightly higher efficiency compared to greater angles which tend to pull on the boundary layer, leading to flow separation.

Supersonic nozzles have many applications in the industry. They are used to control fluid flow, velocity, direction, mass, shape, and/or resulting gas pressure [5]. More complex forms that are often used are parabolic and bell nozzles. Laval nozzles are used for high-speed steam turbines, rocket engines, air turbine drives, gas turbines, thermal ejectors and supersonic aerodynamic tunnels. An under-expanded nozzle evacuates the fluid to a higher outlet pressure comparative with the atmospheric pressure, because the output area is too small for an optimal ratio of the areas [6]. The fluid expansion
is therefore incomplete inside the nozzle and must take place outside of it. The outlet pressure of the nozzle is higher than the local atmospheric pressure. In an over-expanded nozzle, the fluid reaches a lower outlet pressure than the atmospheric, because it has an output area too high for the optimum [7]. If the exit pressure is with 40% less that the atmospheric pressure, a “flow separation” occurs [8], which causes a reduction of the efficiency and also jet instabilities.

The unusual behaviour of the nozzle is influenced by the presence of compression or shock waves within the divergent section; which are powerful compression discontinuities and only exist in the supersonic flow. In a Laval nozzle, at low atmospheric pressure or in vacuum, the Laval nozzle under-expands the exhaust gasses, this effect is leading to losses and reduced efficiency. Bell nozzles are operating with maximum efficiency at design point, at a specific pressure; to obtain desired performances at other altitudes is necessary to make an altitude compromise for the design point [9][17]. A shorter bell shape offers better overall performance due to its much reduced weight, shorter length, lower thrust loss, and only a very small decrease in velocity [10]. The most popular altitude-compensating rocket nozzle to date is the dual bell nozzle, the origin of which dates back to Rocket dyne in the 1950s.

When the combustion chamber has a cross-section greater than four times the critical section area (A1/At > 4), chamber speed v1 may be neglected. However, vehicle space or weight constraints often require smaller areas of the combustion chamber for liquid propulsion engines and design considerations result in small volumes or small perforations or harbour areas for solid propulsion engines. In this case, v1 is no longer neglected, as it helps to increase performance [11].

In the ideal design conditions, the nozzle expansion ratio would be chosen so that the exit pressure matches that of the ambient, so that only a shear layer exists between the high-speed nozzle flow and the ambient (adapted nozzle) [12]. At sea level, the ambient pressure is higher, so that the nozzle flow is over-expanded, and the flow slows down outside of the nozzle through a series of shocks. In more severe cases, the shocks can enter the nozzle, resulting in poor performance. At high altitudes, the flow can be under-expanded and expansion waves deflect the flow outward, resulting in a large “bulge” in the rocket plumes [13].

In this paper, a study regarding the calculation and design of a Laval nozzle, and its influence on the performances obtained is realised. Two geometries are analyzed: one with a cone shape nozzle and the other geometry generated using the characteristics method (bell nozzle).

2. Problem setup

1.1. Mathematical model and geometry

Thermodynamic relations existing in the specific literature underpin the geometry used in this analysis. Two geometries were determined:

- Laval nozzle with a cone shape (conical nozzle) – is the simplest form: chamber area (nozzle inlet), the critical area and the divergent area (nozzle exit)
- Laval nozzle generated using the characteristics method.

The conical nozzle consist of two curves: a convergent curve, designed using a parabolic approximation, and a straight curve with an inflection angle.

The flow of a conical nozzle has a non-axial component, that can cause performances losses. As the cone angle increases, the loss of thrust is also increasing, but for a reduction of weight, large cone nozzles must be used[14]. So the influence of the half angle, can be determined using:

\[ L_{bell} = L_{cone} \cdot \frac{80}{100} \]  

where \( \alpha \) - nozzle divergence half angle, \( \lambda \) - nozzle divergence correction factor, dimensionless.

The length of the bell nozzle is a fraction of the length of a conical nozzle. A reduced bell nozzle length (L_{bell}), is as efficient as a longer 15° conical nozzle (L_{cone}) for the same area ratio [9].
Figure 1 presents the form of the conical and bell nozzle, also some of the parameters required to determine the geometry of a particular type of nozzle.

Some of the thermodynamic relations used to determine flow properties in the important parts of the nozzle are:

\[
T_0 = T_c \left[ 1 + \frac{1}{2} (\gamma - 1) M^2 \right]
\]  

Or

\[
M = \sqrt{\frac{2}{\gamma - 1} \left( \frac{T_0}{T_c} - 1 \right)}
\]

\[
p_0 = p_c \left[ 1 + \frac{1}{2} (\gamma - 1) M^2 \right]^{\gamma/(\gamma - 1)}
\]

where: \( p_0 \) and \( T_0 \) are the pressure and temperature of stagnation; \( M \) – Mach number; \( \gamma \) - ratio of specific heats.

Mass flow through a nozzle is proportional with the throat area \( A_t \) and the chamber pressure \( p_c \), also is inversely proportional with the square root of the ratio between temperature and molar mass of gas \((T/M)\) and gas properties.

\[
\dot{m} = A_t p_c \gamma \sqrt{\frac{2/(\gamma + 1)}{\gamma R T_c}}\]

The geometry was calculated for a chamber pressure \( p_c = 20 \) bar, and a chamber temperature \( T_c = 723K \). At the domain exit, atmospheric conditions are used: \( p_0 = 1.0135 \) bar, \( T_0 = 288 \) K.
The geometrical parameters of the three types of nozzles are presented in table 1.

### Table 1. Geometrical features of the nozzle.

| Dimension                  | Value                  |
|----------------------------|------------------------|
| Convergent diameter        | $D_{\text{convergent}} = 0.006 \text{ [m]}$ |
| Throat diameter            | $D_{\text{throat}} = 0.003 \text{ [m]}$      |
| Exit diameter              | $D_{\text{exit}} = 0.00516 \text{ [m]}$       |
| Length of the converging section | $L_{\text{conv}} = 0.003 \text{ [m]}$  |
| Length of the divergent section – a) | $L_{\text{divergent}} = 0.00404 \text{ [m]}$ |
| Nozzle length – a)         | $L_{\text{nozzle}} = 0.00704 \text{ [m]}$    |
| Length of the divergent section – b) | $L_{\text{diverg}} = 0.003232 \text{ [m]}$ |
| Nozzle length – b)         | $L_{\text{nozzle}} = 0.006232 \text{ [m]}$   |
| External domain length     | $L_{\text{external\_domain}} = 0.06592000 \text{ [m]}$ |
| Height of the external domain | $h_{\text{external\_domain}} = 0.01548 \text{ [m]}$ |

1.2. Spatial discretization

For the analysis of this paper, the commercial software ANSYS CFX has been used. The grid generated is unstructured, and generated using ANSYS ICEM CFD, with a finer resolution on the nozzle area, figure 2. A number of 0.75 million nodes and 3.98 million elements have been obtained for the entire domain.

![Figure 2. Nozzle geometry.](image)

![Figure 3. Grid structure for bell nozzle.](image)
1.3. Boundary conditions
A turbulence model that provides very good results in the case of wall-bound flows even with very separate regions [15] is the SST (Shear Stress Transport), which was used for this numerical simulation.

One of the main gases used for cold gas thrusters is nitrogen, which is playing an important role for in the propulsion of small satellites. A cold gas thruster is the simplest form of a thermal rocket, which has thermal energy as thermal source. This type of gas is preferred because of its performances and storage density. Another important characteristic is the low molecular weight, and also a great specific impulse.

Figure 4 shows the computational field and the regions where boundary conditions are established. Boundary conditions used:
  - INLET: convergent nozzle
  - OUTLET: computational domain: *supersonic*
  - NOZZLE: *wall*
  - EXTERIOR DOMAIN: *opening*:
    - opening pressure and direction: 0 bar
    - opening temperature: 288 K

| Type of condition | Value                              |
|------------------|------------------------------------|
| Inlet velocity   | 78 m/s                             |
| Mass flow        | 0.02 kg/s                          |
| Total temperature| 15°C (288K); 50°C(323K); 450°C (723K) |

Table 2. Geometrical features of the nozzle.

3. Results
The most important parameters for a nozzle are the Mach number, outlet velocity and the thrust force obtained. Figure 5 presents the velocity on the middle of the domain, along its length. An increase of the inlet temperature, is leading to a rise of the Mach number. Mach diamonds, form on the supersonic exhaust gas/plume. These types of shock diamonds are visible due to sudden changes in pressure and density; the static pressure of the gases exiting the nozzle is much higher than the ambient air pressure. Due to the difference between ambient pressure and outlet pressure, the nozzle is under expanded. In all three cases, at least three Mach diamonds are form after the nozzle exhaust. If the gas used is ideal and inviscid, this pattern of Mach disk would repeat indefinitely [16]. The normal shock waves are formed outside the nozzle, due the divergent angle which causes the intersection between the reflected shock wave and the separation shock wave.

In figure 6c) it can be seen that the value of Mach number is higher than 3, and the distance between the Mach disks cannot be seen clearly.
The static pressure illustrated in figure 6, presents the regions with the Mach disks, and the pressure distribution in those areas. The magnitude of those disks is decreasing as the outlet pressure is reaching the atmospheric pressure.

Figure 5. Mach number for the conical nozzle.

The value of Mach number is appropriated for the two cases, conic and bell. The magnitude of the shock diamonds is lower in the case of bell nozzle, figure 7. However, the decrease in intensity is not sufficient to make their number decrease.

Figure 6. Static pressure for the conical nozzle.
The space between the nozzle and the first shock diamond is called “zone of silence” [18], and can be approximated by:

$$x = 0.67D_{exit} \sqrt{\frac{P_0}{P_e}}$$  \hspace{1cm} (7)

Using this approximation, the distances obtained for all cases are found in table 3.
Table 3. Distance between nozzle and first shock diamond.

| Temperature (°C) | Conic  | Bell   |
|------------------|--------|--------|
| 288 | 0.0109 [m] | 0.0104 [m] |
| 323 | 0.0112 [m] | 0.0110 [m] |
| 723 | 0.0149 [m] | 0.0145 [m] |

With the increase of the inlet temperature an increase in distance, between the nozzle and first Mach disk is observed. In the case of the conic nozzle, a rise of temperature with 35K, leads to a distance by about 0.0003 [m]. For the same rise of temperature, the distance between the nozzle and first Mach disk is twice the value obtained for the conic nozzle, 0.0006 [m]. Given that this is a micro-thruster, the distance increase is significant and may influence the safe assembly on the satellite chassis.

Evolution of the velocity in the X direction is presented in figure 9. The lower values are obtained for the bell nozzle, for 288 K and for 323 K. The pattern of velocity evolution is the same for all cases; also near the end of the domain a value of velocity is quite close, ranging around 400 m/s.

Near the symmetry axis, the values of velocity are the same, figure 10. The bell nozzle has the highest velocity for all three cases.

Figure 9. Velocity distribution along the domain length – X direction.
The Mach number obtained at the nozzle exit differs with the change in the inlet temperature, and with the nozzle shape. For the conical shape, the values of the Mach number are with about 0.035 higher than in the case of the bell one, figure 11.

Figure 12 presents a comparison between the forces obtained for all six cases, on the nozzle exit area. The bell nozzle is reaching the greatest value in the case of the bell nozzle, tested for an inlet pressure of 723K. For these case the difference in the geometries, conical and bell, is of 3.95 %. For the other cases, (323 K and 288 K), the difference between them is nearly 1.5%.
4. Conclusions
This paper studies the influence of the geometry on the performances of a cold gas nozzle. Three cases where studied, for two types of geometries: conical nozzle and bell nozzle. An important parameter varied in this study is the inlet temperature. It was confirmed that the decrease of the inlet temperature, leads to a reduction on the nozzle performances. Even if the value of the Mach number in the case of the conical nozzle has higher values that the bell nozzle, overall the bell nozzle for the design conditions has better performances. Between the two geometries, the design parameters play an important role in determining the highest performances, and the nozzle which can reach it; for this study, bell nozzle is the representative nozzle.

5. References
[1] Baloni B D, Kumar S P and Channiwala S A 2017 Computational Analysis of Bell Nozzles, Proceedings of the 4th International Conference of Fluid Flow, Heat and Mass Transfer (FFHMT'17) (Toronto, Canada – August 21 – 23, 2017)
[2] Östlund J 2002 Flow Processes in Rocket Engine Nozzles with Focus on Flow Separation and Side-Loads Technical reports from Royal Institute of Technology Department of Mechanics S-100 44 (Stockholm, Sweden)
[3] Yumus M and Eyi S 2012 Design optimization of rocket nozzles in chemically reacting flows, Computers & Fluids 65 25–34
[4] DeHoff K W and Hrusch N J 2015 Design of the Structural and Propulsion Systems for the 2015 University of Akron Rocket Team. Honors Research Projects 52 (Ohio: University of Akron Main Campus)
[5] Sharma V, Shankar R and Sharma G 2011 Calculation of velocity & Temperature profile of Rocket Nozzle using CFD in zero atmospheric Condition (Dehradun: Bachelor of Technology (Aerospace Engineering) From University of Petroleum and Energy Studies Dehradun)
[6] http://exrocketman.blogspot.ro/2012/12/ramjet-cycle-analyses.html. Retrieved 23 January 2018
[7] Hussaini A A 2013, Isentropic Flow in Converging Nozzles, Mechanical Department/Aeronautical Branch, Chapter 6
[8] Lárusson R 2017 Modal Analysis of Supersonic Flow Separation in Nozzles, Thesis for the degree of doctor of philosophy in thermo and fluid dynamics (Gothenburg: Department of Applied Mechanics Division of Fluid Dynamics, Chalmers University of Technology)
[9] Meyers J M, Over- and Under-expanded Nozzles and Nozzle Configurations, ME 239: Rocket Propulsion, University of Vermont Mechanical Engineering

Figure 12. Force obtained for the cases studied.
[10] PWR Engineering: Nozzle Design. Retrieved 22 January 2018
[11] Sutton G P 2001 Rocket Propulsion Elements: An Introduction to the Engineering of Rockets (7th ed.) (New York: John Wiley&Sons)
[12] Taylor N, Steelant J and Bond R 2011 Experimental comparison of Dual Bell and Expansion Deflection Nozzles, 47th AIAA Joint Propulsion Conference & Exhibit (1 July - 03 August 2011, San Diego, California)
[13] Rao G V R 1958 Exhaust Nozzle Contour for Optimum Thrust Journal of Jet Propulsion 28(6) 377-382
[14] Young R B 2012, Automated Nozzle Design through Axis-Symmetric Method of Characteristics Coupled with Chemical Kinetics (Graduate Thesis, Auburn, Alabama USA)
[15] Georgiadis N J and Yoder D A 2006 Evaluation of Modified Two-Equation Turbulence Models for Jet Flow Predictions (44th AIAA Aerospace Sciences Meeting and Exhibit) (9 - 12 January 2006, Reno, Nevada)
[16] Scott J, Shock Diamonds and Mach Disks, 17 April 2005. Retrieved 26 February 2018
[17] Dragan V 2012 The effect of a thin jet over a concave surface – the development of a new high altitude rocket nozzle, 16th International Conference of Modern Technologies, Quality and Innovation, ModTech 2012: New face of TMCR, Book Series: International Conference ModTech Proceedings pp 317-320
[18] Niessen W M A 1999 Liquid chromatography-mass spectrometry (New York: CRC Press) p 84