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

Design and analysis of a rocket C-D nozzle

Jayaprakash P
Assistant Professor, Department of Aeronautical Engineering, Dhanalakshmi Srinivasan College Of Engineering & Technology - Mamallapuram
Email: prakashair11@gmail.com

Dhinarakaran
Assistant Professor, Department of Mechanical Engineering, Dhanalakshmi Srinivasan College Of Engineering & Technology - Mamallapuram
Email: dhinakaran.mech@dscet.ac.in

Durlab Das
Assistant Professor, Department of Aeronautical Engineering, Dhanalakshmi Srinivasan College Of Engineering & Technology - Mamallapuram
Email: durlab1993@gmail.com

***Abstract*---**The design is based on previous designs used as a benchmark for ours. Since the rocket propulsion for various purposes such as manned missions or launching satellites into orbit are increasing, a nozzle is an integral part that deserves attention. Keeping that clearly in mind, a basic design is done using the knowledge of compressible flow. The values of temperature, pressure and velocity is needed at every section of the nozzle so as to design the shape, insulation and cooling arrangements for the nozzles and the thrust for the same is calculated. Using the design, theoretical calculations are done and CFD simulations are also done. The test result for the both methods is obtained by simulation and theoretical calculation of the flow. In addition, the calculations and the results obtained from using ANSYS Fluent software and Theoretical calculations are compared under the basis of their values. The validation of these formulae is carried out using the Computational Fluid Dynamics (CFD) software ANSYS Fluent.

***Keywords*---**Computational Fluid Dynamics, ANSYS Fluent, Convergent-Divergent.

Introduction

The nozzle may be thought of as a device that converts enthalpy into kinetic energy with no moving parts. A nozzle is used to give the direction to the gases

Manuscript submitted: 18 Feb 2022, Manuscript revised: 27 April 2022, Accepted for publication: 9 June 2022

coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy.Nozzles were invented mainly to changethecharacteristicoftheflowsuch as an increase in pressure or velocity. In 1890 Swedish engineer and inventor Karl Gustaf Patrik de Laval developed a convergent-divergentnozzle that had the capacity to increase a steam jet to a supersonic speed. Since then this nozzle was called as de Laval nozzle and after that this type of nozzles are used for rocket propulsion.

An American engineer Robert Goddard was the first person to integrate a de Laval nozzle in connection with a combustion chamber, increasing the efficiency and achieving supersonic speeds even up to Mach 7. The main uses for a de Laval nozzle fall are in rocket propulsion; but also, there is an increase in the use of the supersonic nozzle in other areas. The United States Air Force has is using rocket nozzles to apply high velocity particles, which are a combination of metals, ceramics, and polymeric materials, over the surfaces of weapon systems.[1] As the use of de Laval nozzles in rocket design have become more important, the parameters of the nozzle are also important. Several research papers and works are conducted to optimize the nozzle to meet certain criteria more effectively. Due to the specific optimization of this project, this survey is focused on de Laval nozzle simulation and optimization.

In 2012 Karla Quintano published a master thesis that detailed work in adjusting the shape parameters of the de Laval nozzle in order to find an optimal setup for making the gas flow exiting the C-D nozzle more uniform. Several software programs were used for the work. A FORTRAN code was used to develop 40 different nozzle shapes. ANSYS and mode Frontier were used to optimize specified parameters of the shapes, and to run simulations on flow and heat transfer. The thesis results showed that the shape of the nozzle had a significant impact on exit flow formation.[2]Jean-Baptiste Mbuyamba published a study regarding nozzle design for a cold gas dynamic spray. While not directly related to rocket nozzles, the study considers de Laval nozzles for its design. It also describes several theoretical elements regarding compressible gas flow in a convergent-divergent nozzle as well as methods to simulate and calculate specific parameters.[3] The process of optimization began with a detailed literary research in order to find all relevant information on supersonic nozzles. This project had a large emphasis on the behavior of flow consisting primarily of compressible fluids. Modeling the flow of air as an ideal cold gas was done using a variety of software platforms such as ANSYS fluent, and a LOCI FORTRAN program. Furthermore, a hot-gas application was modeled following the exact testing conditions as the cold gas while utilizing software programs to analyze different fluid gradients such as pressure, temperature, density, and velocity. A Computer Aided Engineering (CAE) program, mode Frontier, was run to optimize the supersonic nozzle using a variety of geometric configurations.[4]. De Laval nozzle was invented by Gustaf de Laval, a Swedish inventor. This type of nozzles is most commonly used in supersonic vehicles and rockets. The type of nozzle is convergent-divergent nozzle, which is employed to provide supersonic velocity flows at the nozzle exit. In this journal, the theoretical analysis of de Laval nozzle is carried out by formulating required

nozzle equations and verifying the results by using the CFD software (ANSYS FLUENT).

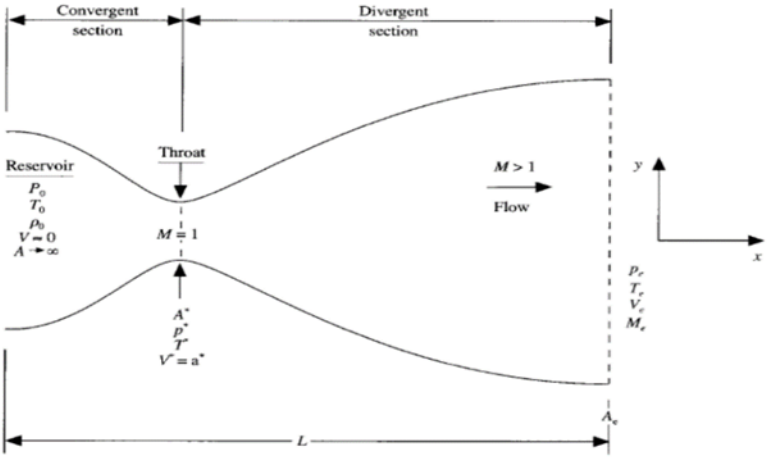


Fig.1.5-De Laval nozzle

First, we will theoretically calculate the velocity, temperature and pressure at different cross sections of the nozzle using the formulae and then the results are verified by using simulation software. This provides a large number of advantages as we can able to visualize the flow in the nozzles.

Nozzle parameters

The flow through the de Laval nozzle is a compressible flow for higher Mach numbers but the compression of flow is reversible. By second law of thermodynamics reversible process has constant entropy thus the flow through the de Laval nozzle is an isentropic flow. The following equations govern the isentropic process. The gases that exit the combustion chamber is different in composition than that is taken by the compressor, The exhaust gas is composed of several other constituents like water, carbon monoxide, hydrocarbons, nitrous oxides and sulfur dioxides therefore the 'γ' ratio of specific heats takes a different value and has to be considered during the analysis.

With an increased emphasis on rocket propulsion including the founding of notable private space agencies and a noticeable increase in the number of launches of satellites and other devices into space; improvements in rocket propulsion are a necessary component in making the future of space exploration more efficient. Specifically, in regards to the profile, geometry, and pressure distribution inside the nozzle, each application will have its own specific optimized shape. These improvements, namely how pressure is distributed throughout the nozzle will indicate whether the highest thrust is being achieved, minimal boundary layer separation, and best flow properties are being achieved for any given geometric shape which is being optimized for a given function.

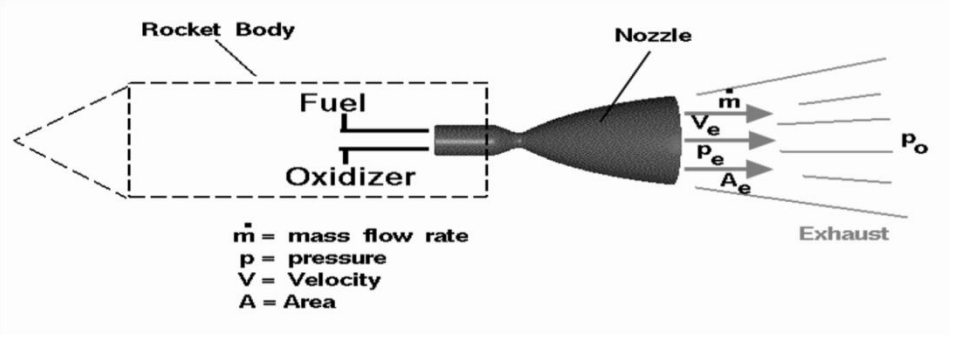
Table 1.1 Nozzle Dimension

Parameter	Dimension
Total Nozzle Length (mm)	484
Inlet Diameter (mm)	166.6
Throat Diameter (mm)	34.5
Outlet Diameter (mm)	179.0
Chamber Length (mm)	99.93
Convergent Angle (deg)	35
Divergent Angle (deg)	11.25
Throat Radius of Curvature (mm)	70
Convergent Radius of Curvature (mm)	40

These enhancements in the pressure distribution conditions inside the nozzle will optimize the given propulsion system for whatever its given operating conditions will be. Simultaneously, greatest efficiency will be achieved while also saving on the cost from losses associated with previous less efficient designs. The outcome is an expected maximization in efficiency while minimizing resources, such a combination must be achieved if progression of private space enterprise is to be sustained and expanded through the approaching decades.

Methodology

Rocket thrust equation



Thrust (F) =$\dot{m}$ V_e + (P_e-P_o) A_e

Fig1.6-Rocket Thrust Equation

Where,

$$Mach\ number = \frac{V}{A}$$

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

$$\frac{p}{\rho^{\gamma}} = c$$

Computer drawing of a convergent-divergent nozzle with equations that describe the operation:

Ramjets, scramjets, and rockets all use nozzles to accelerate hot exhaust to produce thrust as described by Newton's third law of motion. The amount of thrust produced by the engine depends on the mass flow rate through the engine, the exit velocity of the flow, and the pressure at the exit of the engine. The value of these three flow variables are all determined by the nozzle design.

A nozzle is a relatively simple device, just a specially shaped tube through which hot gases flow. Ramjets and rockets typically use a fixed convergent section followed by a fixed divergent section for the design of the nozzle. This nozzle configuration is called a convergent-divergent, or CD, nozzle. In a CD nozzle, the hot exhaust leaves the combustion chamber and converges down to the minimum area, or throat, of the nozzle. The throat size is chosen to choke the flow and set the mass flow rate through the system. The flow in the throat is sonic which means the Mach number is equal to one in the throat. Downstream of the throat, the geometry diverges and the flow is isentropically expanded to a supersonic Mach number that depends on the area ratio of the exit to the throat. The expansion of a supersonic flow causes the static pressure and temperature to decrease from the throat to the exit, so the amount of the expansion also determines the exit pressure and temperature. The exit temperature determines the exit speed of sound, which determines the exit velocity. The exit velocity, pressure, and mass flow through the nozzle determines the amount of thrust produced by the nozzle.

On this slide we derive the equations which explain and describe why a supersonic flow accelerates in the divergent section of the nozzle while a subsonic flow decelerates in a divergent duct. We begin with the conservation of mass equation:

$$\dot{m} = r * V * A = \text{constant}$$

where $\dot{m}$ is the mass flow rate, r is the gas density, V is the gas velocity, and A is the cross-sectional flow area. If we differentiate this equation, we obtain:

$$V * A * dr + r * A * dV + r * V * dA = 0$$

divide by (r * V * A) to get:

$$dr / r + dV / V + dA / A = 0$$

Now we use the conservation of momentum equation:

$$r * V * dV = - dp$$

and an isentropic flow relation:

$$dp / p = \gamma * dr / r$$

where γ is the ratio of specific heats. This is Equation on the page which contains the derivation of the isentropic flow relations. We can use algebra on this equation to obtain:

$$dp = \gamma * p / r * dr$$

and use the equation of state

$$p / r = R * T$$

where R is the gas constant and T is temperature, to get:

$$dp = \gamma * R * T * dr$$

$\gamma * R * T$ is the square of the speed of sound a:

$$dp = (a^2) * dr$$

Combining this equation for the change in pressure with the momentum equation we obtain:

$$r * V * dV = - (a^2) * dr$$

$$V / (a^2) * dV = - dr / r$$

$$- (M^2) * dV / V = dr / r$$

using the definition of the Mach number $M = V / a$. Now we substitute this value of (dr /r) into the mass flow equation to get:

$$- (M^2) * dV / V + dV / V + dA / A = 0$$

$$(1 - M^2) * dV / V = - dA / A$$

This equation tells us how the velocity V changes when the area A changes, and the results depend on the Mach number M of the flow. If the flow is subsonic then ($M < 1$) and the term multiplying the velocity change is positive ($1 - M^2 > 0$). An increase in the area ($dA> 0$) produces a negative increase (decrease) in the velocity ($dV< 0$). For our CD nozzle, if the flow in the throat is subsonic, the flow downstream of the throat will decelerate and stay subsonic. So, if the converging section is too large and does not choke the flow in the throat, the exit velocity is very slow and doesn't produce much thrust. On the other hand, if the converging section is small enough so that the flow chokes in the throat, then a slight increase in area causes the flow to go supersonic. For a supersonic flow ($M > 1$) the term multiplying velocity change is negative ($1 - M^2 < 0$). Then an increase in the area ($dA> 0$) produces an increase in the velocity ($dV> 0$). This effect is exactly the opposite of what happens subsonically. Why the big difference? Because, to conserve mass in a supersonic (compressible) flow, both the density and the velocity are changing as we change the area. For subsonic (incompressible) flows, the density remains fairly constant, so the increase in area produces only a

change in velocity. But in supersonic flows, there are two changes; the velocity and the density. The equation:

- (M^2) * dV / V = dr / r

tells us that for M > 1, the change in density is much greater than the change in velocity. To conserve both mass and momentum in a supersonic flow, the velocity increases and the density decreases as the area is increased.

The equations used here are for the findings of one-dimensional nozzle flow. It helps in the idealization of full two- or three-dimensional flow equations and real aero thermochemical behaviour.
The Nozzle equation is derived using the continuity equation and steady state energy equations.

The continuity equation is

$$\rho_1 A_1 v_1 = \rho_2 A_2 v_2$$

The steady flow Energy equation is

$$(h_i - h_e) = (V_e^2 - V_i^2)/2$$

The following equations is derived using Continuity equation and Steady flow Energy equation:

$$\frac{A_x}{A_{th}} = \left(\frac{T_{th}}{T_x}\right)^{\frac{1}{\gamma-1}} \times \frac{\sqrt{\gamma R T_{th}}}{V_x} \quad (1)$$

$$C_p \times T_{th} + \frac{V_{th}^2}{2} = C_p \times T_x + \frac{V_x^2}{2} \quad (2)$$

Solving equation (1) and equation (2) simultaneously, we get the values of velocity (V_x) and temperature (T_x) at the required section of the nozzle. Pressure at the section can be calculated using isentropic laws.

Table 4.1-Theoretical results

Section of nozzle	(Ax/Ath)	Velocity(m/s)	Temp.(K)	Pressure(bar)
Convergent	1.6	179.1	3260.2	88.33
Convergent	1.4	499.91	3694.21	75.77
Throat	1	1120.64	2675.89	48.98
Divergent	1.2	1239.68	2456.07	33.80
Divergent	3	2198832	2137.65	11.50

4.3 Theoretical calculation result and comparison with CFD result

From the velocity contour obtained above and the gas dynamics relations, the Mach number for the revised nozzle is M= 2.7

Now by theoretical calculation,

Stagnation pressure P_0= 40 bar

Stagnation temperature T_0= 3677

Inlet Diameter D_1= 10 cm

Exit Diameter D_2= 20 cm
Throat Diameter D_t= 5 cm
Entry Mach number M_1= 0.32
Exit Temperature $T_2 =$ 1468 K

Upon calculations as illustrated above the theoretical calculation of Mach number is M=2.93

The theoretical result is greater than the results by CFD as expected.
Pre-processing of the nozzle made in ANSYS FLUENT.
2-Dimensional and double precision settings were used while reading the mesh. The mesh is scaled down because all dimensions are in mm. The mesh was checked in fluent software and no errors was founded out.And then the next process is carried out which is giving Boundary Conditions.

Table 3.1-Problem setup

General	Solver type: Density-based 2D Space: Axi-symmetric
Models	Energy equation: On
	Viscous model: Standard k-e model, realizable, enhanced wall treatment
Materials	Density: ideal gas Cp = 1880J/kg K Adiabatic index = 1.19 Viscosity= 8.983×10^{-5}Pa. s Thermal conductivity = 0.0142 W/Mk Mean molecular mass = 27.7 g/mol
Boundary conditions	Inlet Pressure = 100bar Inlet Temperature = 3300K Outlet Temperature = 1700K (For initialization purpose only)

Table 3.2 - Solution

Solution controls	Courant number = 5
Solution initialization	Compute from: Inlet
Run calculation	Check case No. of iterations: 2000 Click Calculation

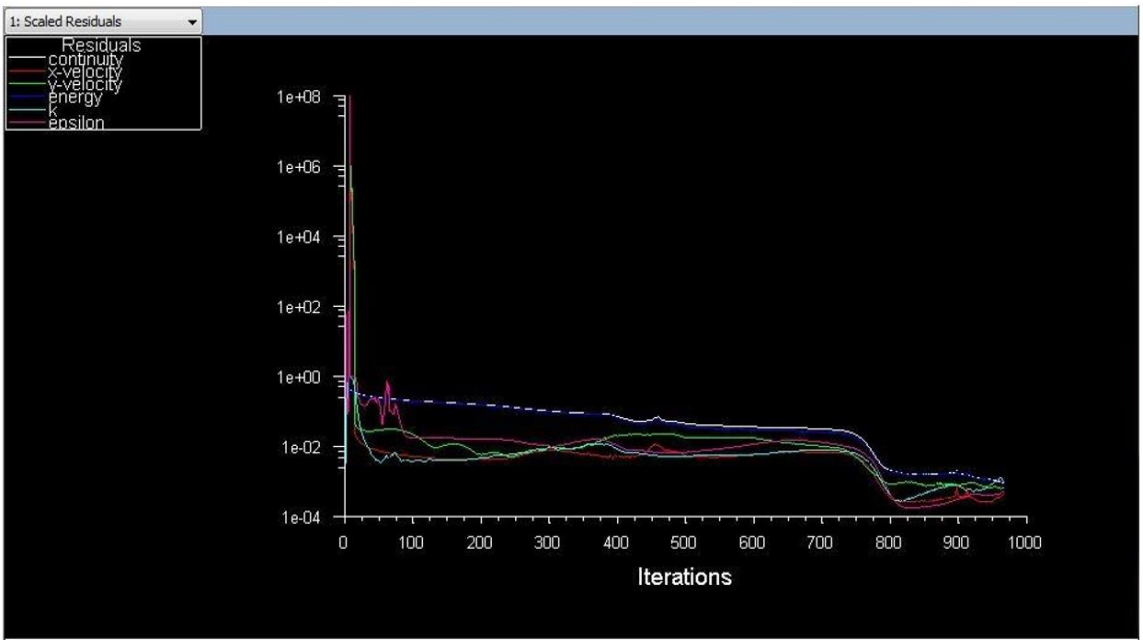


Fig.3.4-Scaled Residuals

The solution was completed after 1243iterations. The value of scaled residuals was lesser than $10e^{-3}$.

Results & Discussion

The results obtained by Computational Fluid Dynamics (CFD) are almost identical to those obtained theoretically. The tables below compare theoretical results to CFD results.
The results are compared by using three contours

1. Velocity Comparison
2. Pressure Comparison
3. Temperature Comparison

6.1 Velocity Comparison

The velocities of the theoretical values and CFD values are not the same but in such a way that they have minute differences in their values but they don't play a major concern because the software will consider the shock waves, drag, boundary layers, frictional forces and so on. But in the case of Theoretical values all we do is neglecting those values.

Table 6.1-Velocity comparison

Section	(Ax/Ath)	Velocity (m/s)	
		Theoretical	CFD
Convergent	1.500	170.25	165.60
Convergent	1.2	482.91	469.32
Throat	1	1032.26	1079.38
Divergent	1.200	1335.28	1321.57
Divergent	4	2048.23	2086.39
Outlet	7.142	2387.52	2400.32

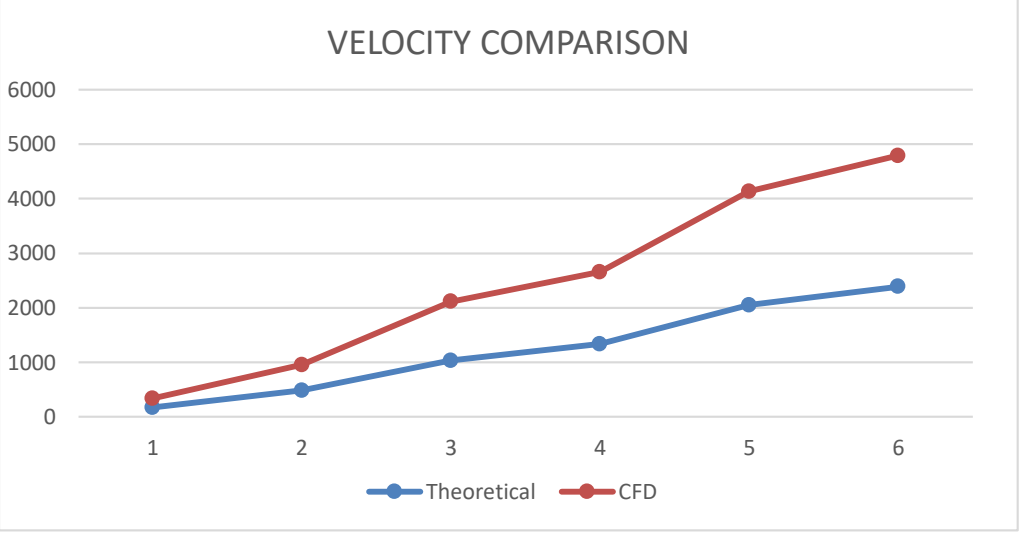


Figure 6.1-Velocity Comparison

6.2 Temperature Comparison:

Table6.2-Temperature comparison

Section	(Ax/Ath)	Temperature (K)	
		Theoretical	CFD
Convergent	1.500	3256.08	3248.90
Convergent	1.2	3269.12	3224.79
Throat	1	2989.98	2945.81
Divergent	1.200	2760.50	2751.30
Divergent	4	2137.56	2070.83
Outlet	7.142	1724.90	1760.89

The Temperature of the theoretical values and CFD values are not the same but in such a way that they have minute differences in their values but they don't play a major concern because the software will consider the shock waves, drag,

boundary layers, frictional forces and so on. But in the case of Theoretical values all we do is neglecting those values.

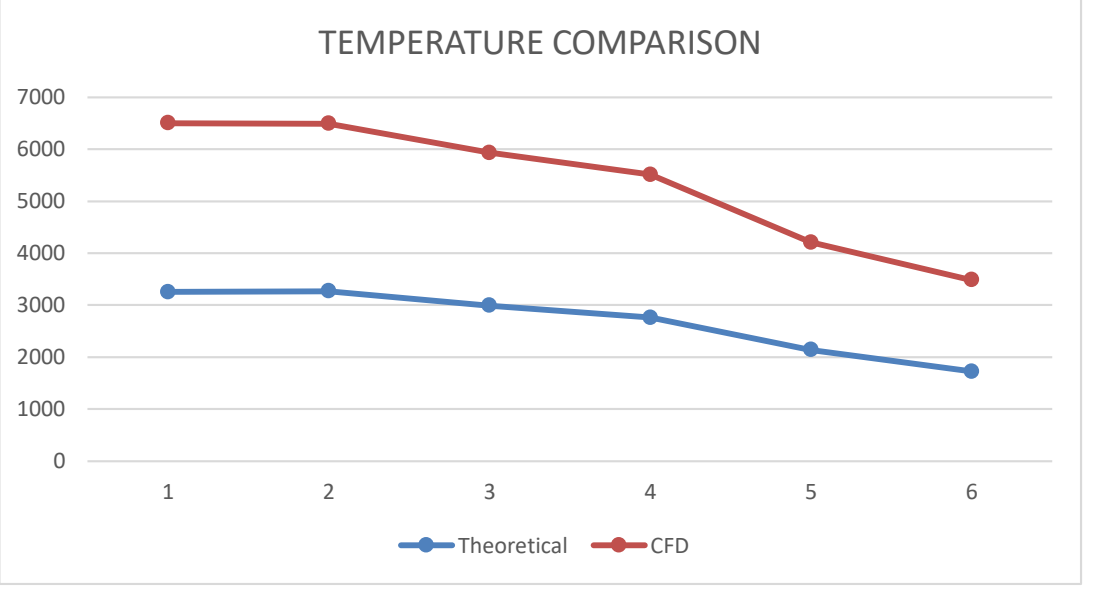


Figure 6.2-Temperature Comparison

6.3 Pressure Comparison:

The Pressure of the theoretical values and CFD values are not the same but in such a way that they have minute differences in their values but they don't play a major concern because the software will consider the shock waves, drag, boundary layers, frictional forces and so on. But in the case of Theoretical values all we do is neglecting those values.

Table6.3-Pressure comparison

Section	(Ax/&)	Pressure (bar)	
		Theoretical	CFD
Convergent	1.500	86.54	93.25
Convergent	1.2	81.65	84.46
Throat	1	50.96	49.53
Divergent	1.200	32.58	36.24
Divergent	4	10.98	7.14
Outlet	7.142	1.84	0.978

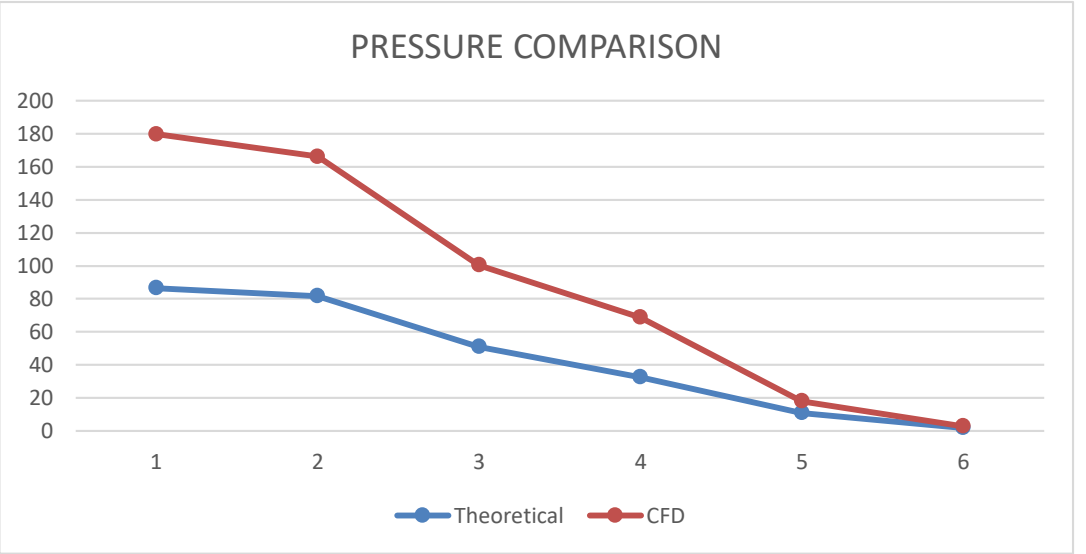


Figure 6.3-Pressure Comparison

Velocity Contours:

The velocity contour is used so as to know the change in velocity from the inlet to the exit. The velocity is minimum at the inlet and goes on increasing till the nozzle exit. The maximum velocity at the throat is Mach 1. This is known as choked flow. The velocity at the nozzle exit is 2498.56m/s.

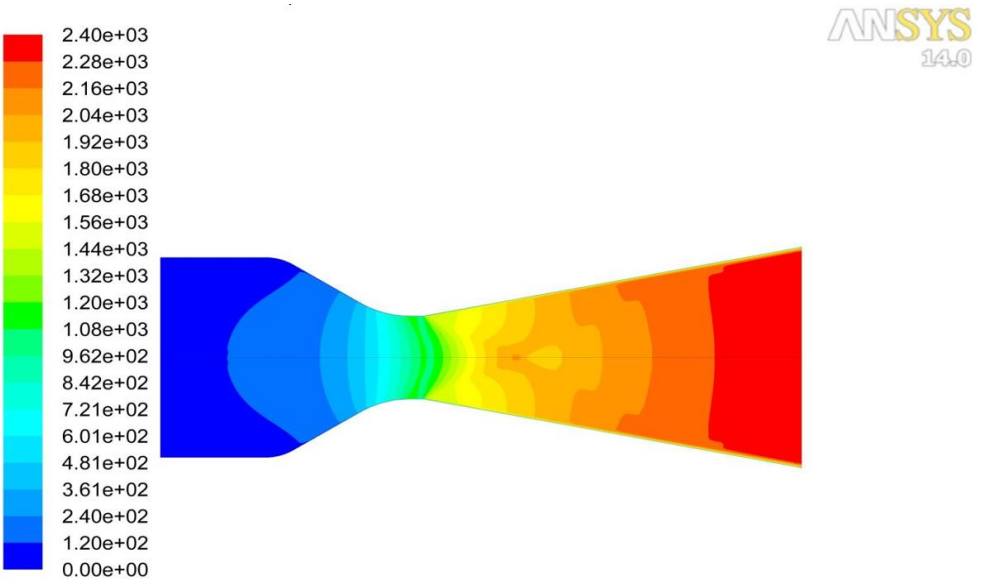


Fig. 5.1 Contours of Velocity Magnitude (m/s)

5.2 Temperature Contours:

The temperature is very important factor which should be considered for the findings of cooling techniques, or else structural deformation occurs at the nozzle. The temperature is maximum at the inlet and minimum at the exit. The maximum temperature at the exit is 1801.8K.

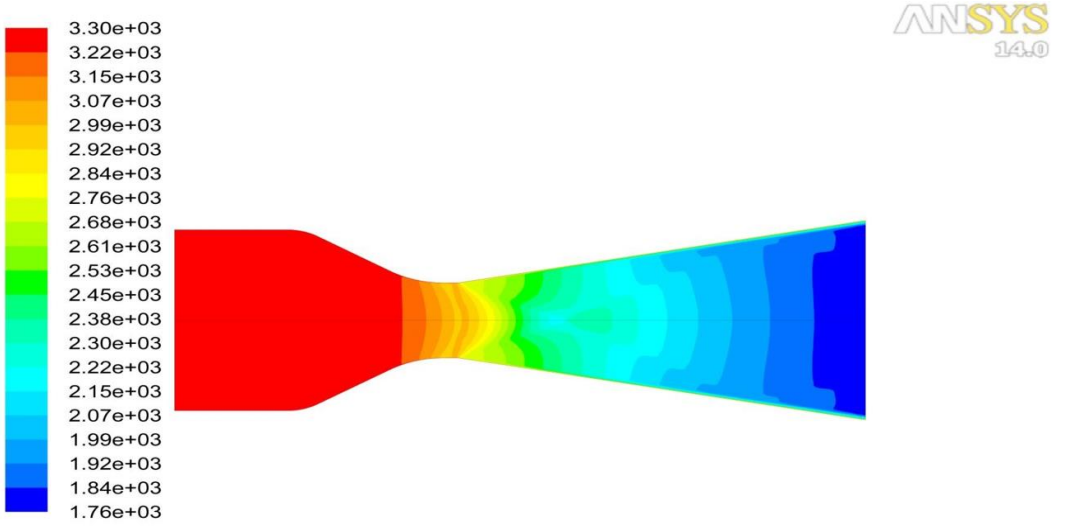


Fig. 5.2 Contours of Static Temperature (K)

5.3 Pressure Contours:

Since the Thrust is majorly dependent on pressure drop at the exit it is necessary that the pressure should be very less for maximum thrust. The pressure is maximum at the inlet and minimum at the exit. The exit static pressure is 0.927 bar. Pressure will be reduced due to the formation of shocks at the Throat.

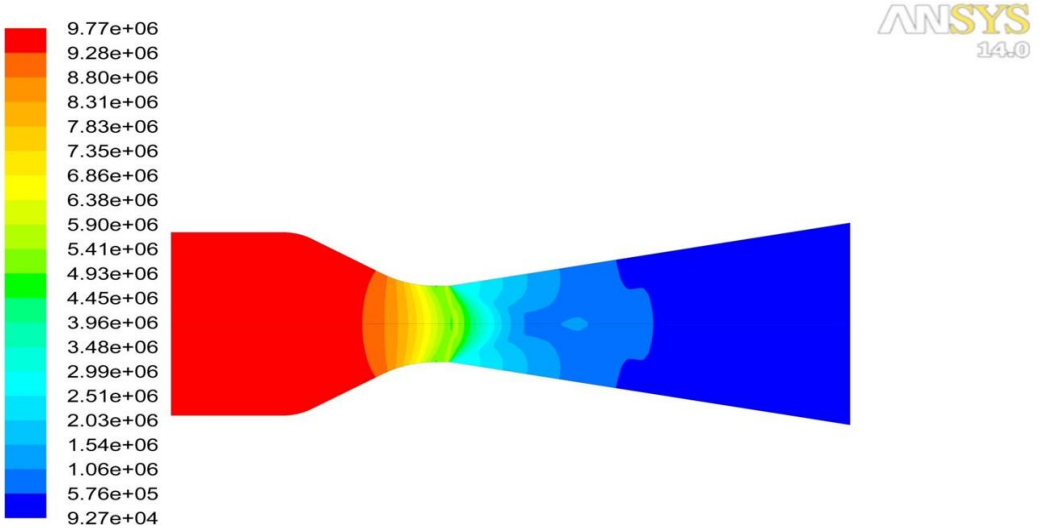


Fig.5.3 Contours of Static Pressure (pascal)

The parameters of the basic design of the nozzle, at the exit, are summarized in a table below

Table 5.2-Nozzle design parameters

Sl.no	Content	Value
1.	Exit density	$0.69 kg/m^3$
2.	Exit pressure	$3.814 \times 10^5 N/m^2$
3.	Exit temperature	1468 K
4.	Exit velocity	2257 m/s^2

Conclusion

The values of CFD and Theoretical methods are slightly different because the CFD software considers the factors like boundary layer effects, shock waves, velocity component and more, therefore, there is difference in the results.The difference in the results of theoretical calculations and CFD are quite small so they can be used for the calculation of the above parameters.From the Result it is clear that the one-dimensional nozzle analysis is enough to find out the nozzle performance therefore giving clarity for the selection of the nozzle.Finally, the three validations of the nozzle are completed and it is clear that the nozzle we designed were more efficient and produces more thrust than the existing nozzles.

References

[1] A comprehensive project report titled “Computational analysis of a supersonic nozzle”, by TEAM18

[2] Quintao, Karla K, “Design and optimization of nozzle shapes for maximum uniformity of exit flow” [2012]. *FIU electronic theses and dissertations.* Paper 779

[3] Mbuyamba, Jean-Baptiste Mulumba, “Calculation and design of supersonic nozzles for cold gas dynamic spraying using MATLAB and ANSYS FLUENT” [2013]

[4] Hagemann, Gerald, Hans Immich, Thong Nguyen and Gennady Dunmov,” Advanced Rocket nozzles”, *Journal of propulsion and power 14.5*(1998).

[5] “Water channel for supersonic flow investigation”, Chemical Engineering and Technology group, Bhabha Atomic Research centre

[6] Ernawati, E., Baso, Y. S., Hidayanty, H., Syarif, S., Aminuddin, A., & Bahar, B. (2022). The effects of anemia education using web-based she smart to improve knowledge, attitudes, and practice in adolescent girls. International Journal of Health & Medical Sciences, 5(1), 44-49. https://doi.org/10.21744/ijhms.v5n1.1831

[7] “Design and optimization of de Laval nozzle to prevent shock induced flow separation”, Aeronautical Engineering Department, Hindustan University George P. Sutton and Oscar Biblarz, “Rocket Propulsion Elements”,

[8] A Wiley- Interscience Publication, Seventh Edition, 2001, (pp 1-99).

[9] K.Ramamurthy, “Rocket Propulsion”, Macmillan publishers India, 2012 edition, (pp 54-89).

[10] K.M. Pandey and S.K.Yadav, “CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1”, Journal of Environmental Research and Development, Vol 5, No 2, 2010, (pp 308-321).

[11] Suryasa, I. W., Rodríguez-Gámez, M., & Koldoris, T. (2022). Post-pandemic health and its sustainability: Educational situation. International Journal of Health Sciences, 6(1), i-v. https://doi.org/10.53730/ijhs.v6n1.5949

[12] Yunus A. Çengel and John M. Cimbala, “Fluid Mechanics”, Tata McGraw-Hill New York, Second edition, (pp 853- 910).