Designing and Numerical analysis of supersonic Converging-Diverging (CD) nozzle

Sumit kumar[1] Raj kumar singh[2]

[1,2]Department of mechanical Engineering, Delhi Technological University, Shahbad Daulatpur, Bawana Road, Delhi, India

Abstract

In this paper, a supersonic Converging-Diverging (CD) nozzle is designed by using method of characteristics for the optimum exit Mach number correspond to desire input condition and a numerical simulation has been performed to for the same nozzle. To design the supersonic CD-nozzle an open-source MATLAB code is used is generate the profile curve coordinates of the diverging wall of the nozzle and then by inserting these coordinates in SOLIDWORKS, a two-dimensional nozzle simulation domain is created. Since, in supersonic flow the, the flow in the converging and diverging section is independent to each other, a random geometry profile is used for converging section. The numerical is performed in the ANSYS (FLUENT) and output parameters have calculated at the exit of the nozzle. An analytical solution is also calculated to validate the result of the simulation by assuming the isentropic flow in nozzle. This study will give an idea about the optimum designing of supersonic CD-nozzle

Keywords; Supersonic, CD-nozzle, Method of Characteristics, Mach Number, Minimum length Nozzle, Gradual increasing Nozzle

1. Introduction

Supersonic CD-nozzle is a simple device having gradual varying cross-section used to accelerate the compressible fluid to a supersonic speed i.e., higher than the speed of sound in the same fluid. CD-nozzle has application in steam turbines to expand the steam and in the aircraft, rocket and satellites to produce high value thrust to propel the vehicle. CD-nozzle converts the high enthalpy energy of a highly pressurized gas into a high kinetic energy and this kinetic energy, according to newton's third law produces a reaction force known as thrust. Although, nozzle is a very simple device, the flow process inside a supersonic nozzle is very complex phenomena and its heavily depends upon the geometry profile of the nozzle. A small change in geometry can significantly alter the flow inside the nozzle, hence the performance of the nozzle and output parameters. The design of the supersonic nozzle should be such that it gives a shock free expansion of the gases in the nozzle to avoid losses and nozzle life deterioration, also in case of nozzle design for rocket its length should be to minimum. As the length of the nozzle increases, the weight associated with it also increases, hence increase in waste of thrust to overcome the weight of the vehicle itself and payload caring capacity of the vehicle reduces. Our main goal is to increase the payload caring capacity of the vehicle; hence the nozzle design should be such that should have minimum length.

Hussain et al. [1] investigated numerical simulation of flow in CD-nozzle having three distinct shape of divergent wall profile i.e., cone shape, bell and triangular. But there is no clarity about process of designing of the nozzle, they just used a random shape for the nozzle. Goyal S. et al. [2] performed the CFD analysis of a simple triangular shaped nozzle by varying the divergent angle from 4.76 to 10 degree and kept all input parameter constant. Mubarak A.K. et al. [3] performed numerical and experimental investigation of the conical, bell and double parabolic nozzle and compared their result. The maximum wall angle for the wall is kept half of the Prandtl-Meyer function and the best performance is shown by the double parabola nozzle than the convectional nozzle. Md Akhtar et al. [4] developed a MATLAB for designing the minimum length nozzle using method of characteristics and established a numerical solution for two dimensional, steady, inviscid, irrotational and supersonic flow. Naveen Kumar k et al. [5] employed a second throat ejector diffuser system to simulate actual ambient vacuum conditions present in the earth orbit where satellites operate. The CFD analysis of the flow is carried out, the pressure, temperature and velocity variation are studied. Also, the design of the diffuser is optimized by the variation of different parameter for the sable and shock free flow at the exit. F. Ferdaus et al. [6] designed and optimized a straight nozzle to attain supersonic flow and to achieve maximum thrust without flow separation due to shock waves. They confirmed that, angle of deflection on the divergent portion increases speed and shock at the exit induces the flow separation. [6]. All of these, researcher have focused either only designing of the nozzle or simulation analysis of the nozzle. So, our objective is to designing a supersonic CD-nozzle as well as numerical and analytical solution for the same, for a comprehensive study.

2. Nozzle Design

The performance of the nozzle is heavily dependent upon the diverging wall profile, and for that the flow at the exit should be parallel to slip line so that, there is no formation of the shock waves which deteriorate the nozzle life and result in increase in loss of energy. Also, the length of the nozzle should minimum as much as possible. To designing the nozzle of the used method of characteristics, which is a popular method of designing nozzle used by even NASA and SpaceX. A nozzle diverging wall has two section, the first one is expansion section, in which flow expands and result in increase in the Mach number of the flow. This expansion of the flow is takes place when flow crosses the expansion waves or expansion fan originate over the length of this expand section of the wall, and result in, increase in Velocity, Mach number and decrease in the pressure of the fluid, also there is change in the direction of the flow particle or fluid particle. Now, as we know, the direction of the flow at the exit of the nozzle should parallel, then the second section of the wall, known as straightening section comes into the picture and ensure that the flow at the exit should be parallel. In the above nozzle, since the expansion of fluid is occurring over a length of nozzle, this nozzle is known as gradual expansion nozzle. The second kind of nozzle in which, the length of the expansion wall is reduced to a point, a sharp corner point at the downstream of the throat, at this point the diverging wall has maximum wall angle it gradually reduces to zero at the exit. This type of nozzle known as the minimum length nozzle, because it only has straightening section and expansion of flow occur through a centralized expansion waves originates form sharp corner. The minimum length nozzle is used in the rocket, because it has the minimum length that can be possible of the nozzle that can be designed using the method of characteristics.

The concept of the designing of the wall of nozzle is such that, there should be no reflection of the expansion waves once it impinges on the wall and this can be achieved by making the wall angle equal to direction of the flow near by the wall. The supersonic flow is characterized by the mesh generated by the centralized expansion fan or Prandtl waves. We calculated the point of the wall by using the compatibility equation for the characteristics line and the other Prandtl-Meyer equation, all these equations are available in the any standard book for modern compressible flow. In this paper we used an open-source MATLAB code based on method of characteristics for calculating the contour point of the nozzle wall for the design problem that is given below in table;

Parameters	**Value**
Chamber pressure	2268000 Pa
Chamber temperature	1200K
Thrust (N)	4000
Altitude (m)	7500
Coefficient of heat (γ)	1.4
Gas characteristic constant for air (R)	*287* KJ/KgK
Throat radius	35mm

Table1 Design problem parameters for the nozzle design

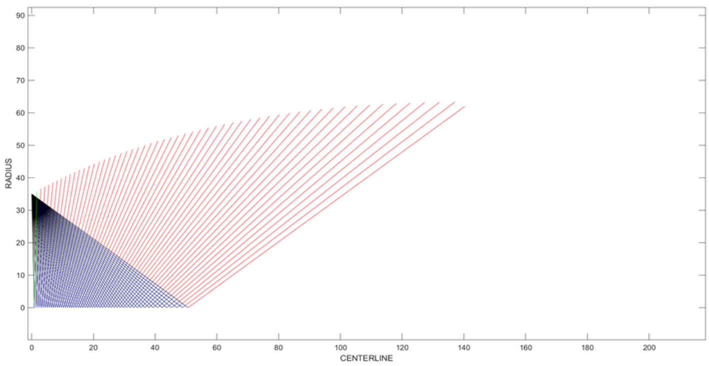

Fig.1 Expansion wave curves and nozzle wall contour (MATLAB)

The designed nozzle has value of exit radius is 6*7.87*mm, length of nozzle is 137.04mm and having maximum wall angle 27.76 degrees.

3. Computational simulation of nozzle

For the computational simulation ANSYS software has been used, which is used by all over the world for academic purpose. The state of the fluid flowing in the nozzle is describe by three parameters like velocity, pressure, and temperature. The fluid flowing through the nozzle must have to satisfy the governing equations that describes the flow through the nozzle, like mass conservation or continuity equation, momentum conservation equation, energy equation and gas state equation. For the analysis of the compressible fluid, we used density-based solver and the model is used is the standard k-epsilon turbulence model is used which has been used by many of the researcher in their study. The fluid is air as Ideal gas and Sutherland model has been used for the viscosity.

3.1 Governing equations involved

1. Mass Conservation Equation or Continuity Equation

$$\frac{d\rho}{\rho} + \frac{dA}{A} + \frac{du}{u} = 0$$

2. Conservation of Momentum

$$\frac{dP}{\rho} + udu = 0$$

3. Energy Equation

$$dh + udu = 0$$

4. Equation of state

$$\frac{dp}{p} - \frac{d\rho}{\rho} - \frac{dT}{T} = 0$$

5. k-ε Equation

$$\frac{\partial \rho \kappa}{\partial t} + div(\rho u \kappa) = div\left[\left(\mu_t + \frac{\rho \mu_t}{\sigma_\kappa}\right) grad\kappa\right] + \rho \mu_t G - \rho \varepsilon$$

$$\frac{\partial \rho \varepsilon}{\partial t} + div(\rho u \varepsilon) = div\left[\left(\mu_t + \frac{\rho \mu_t}{\sigma_\varepsilon}\right) grad\varepsilon\right] + C_{1\varepsilon} \rho \mu_t \left(\frac{\varepsilon}{\kappa}\right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{\kappa}$$

3.2 Meshing

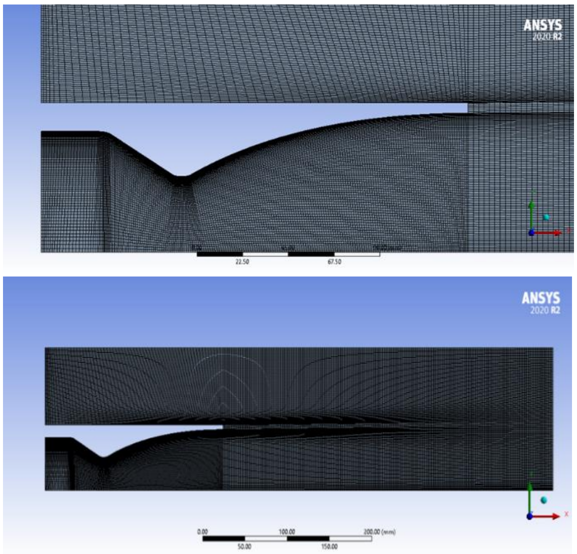

Fig. 2 CD-nozzle domain meshing in ANSYS (Fluent)

The number of nodes and elements in the meshing of the nozzle simulation domain are;

Nodes = 45064   Elements = 44484

3.3 Setup and Solution

Parameter	**Value**
Inlet Pressure	2268000 Pa
Inlet Temperature	1200 K
Outlet pressure	39365 Pa
Outlet Temperature	243K

Table2 List of input parameter in numerical simulation

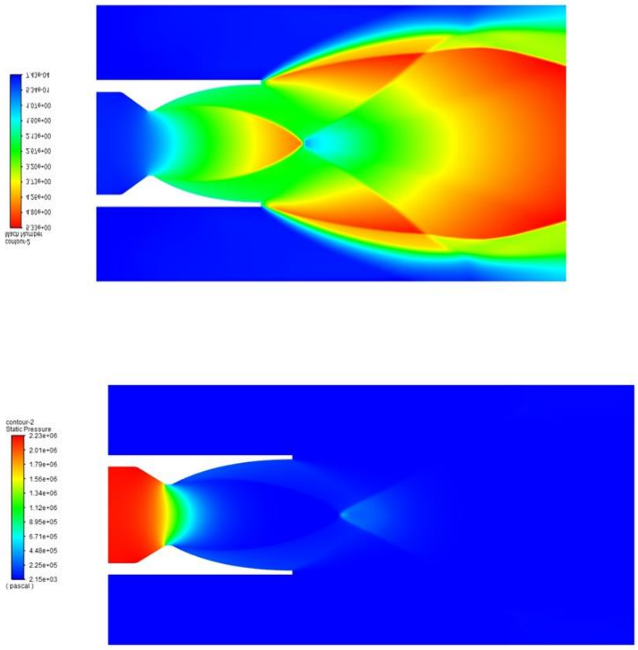

Fig.3 Variation of Mach number and static pressure in the complete simulation domain (ANSYS)

3.4 Result

Graph for different parameter value has been plotted at the exit of the nozzle

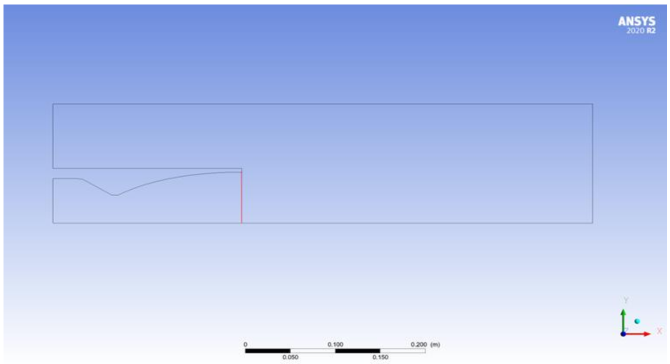

Fig. 4 Calculation at the exit of nozzle (ANSYS)

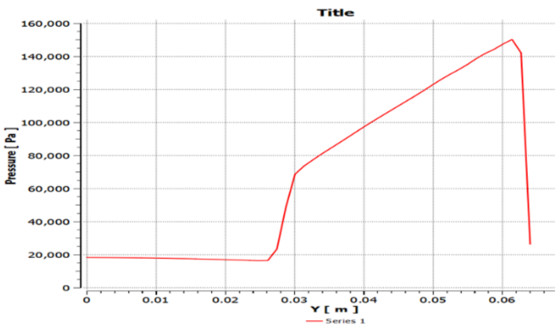

Fig.5 Pressure variation from centreline to nozzle wall at exit (ANSYS)

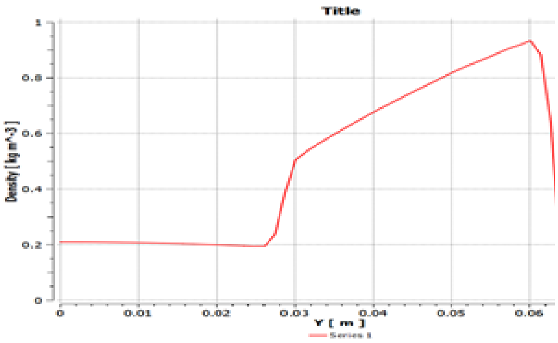

Fig. 6 Density variation from centreline to nozzle wall at exit (ANSYS)

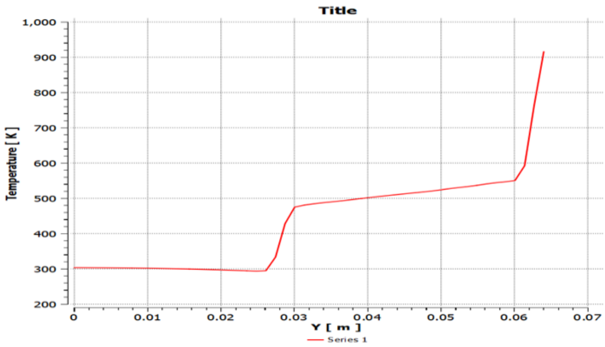

Fig. 7 Temperature variation from centreline to nozzle wall at exit (ANSYS)

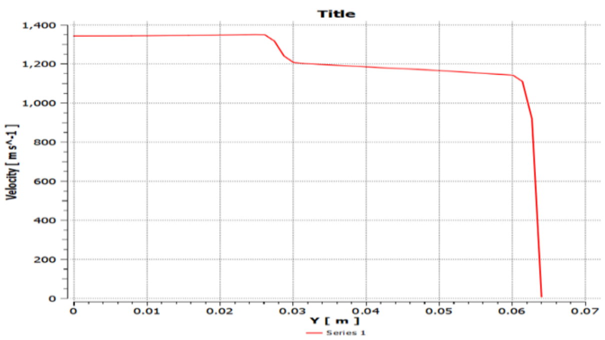

Fig. 8 velocity variation from centreline to nozzle wall at exit (ANSYS)

3.5 Post processing to plot graph of Mach number, Mass flow rate and Thrust at nozzle exit

We had calculated the variation of the Mach number, mass flow rate and thrust value from centreline to wall at the exit of the nozzle in EXCEL, from the data obtained from the numerical simulation in ANSYS (FLUENT).

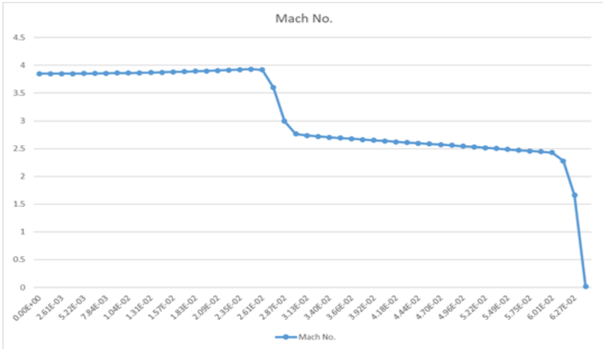

Fig. 9 Variation of Mach number from centreline to wall at the exit (EXCEL)

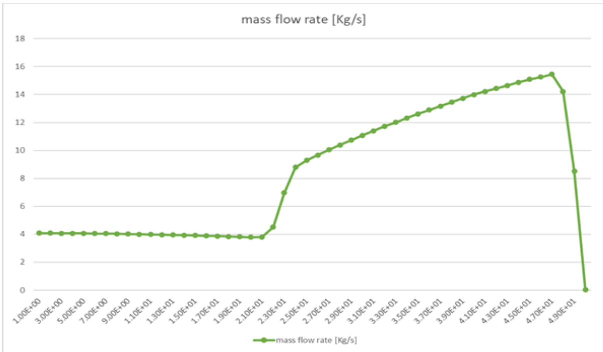

Fig. 10 Variation of Mass flow rate from centreline to wall at the exit (EXCEL)

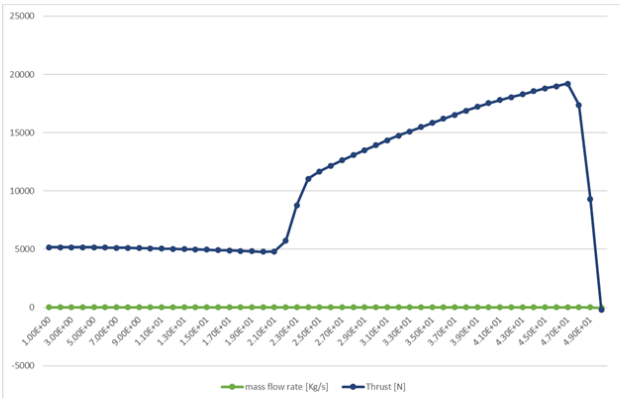

Fig. 11 Variation of Thrust value from centreline to wall at the exit (EXCEL)

In numerical simulation, we have to ignore the beginning and the ending values of the curve, because of the shock at the boundary and at the centreline, the average value from table is taken at the vertical distance from the centreline is .035m. the corresponding average values of the Mach number is 2.69, mass flow rate is 10.3925Kg/s and the average thrust is 13073.4854 N

4. Analytical solution

For the validation of the result obtained from the computational methodology, we will calculate the thrust value analytical assuming isentropic flow.

Parameter	Value
Inlet Pressure	2268000 Pa
Inlet Temperature	1200 K
Outlet pressure	39365 Pa
Outlet Temperature	243K
Throat radius (area)	35mm or .035m (0.00384845 m^2)
Exit Radius (area)	67.87mm or 0.06787m (0.0144712 m^2)
Area ratio	10.269277
R	287 KJ/KgK

Table3 Design parameters

The calculations for the various nozzle parameters i.e., velocity, pressure, density, temperature and Mach number has been performed by hand by assuming isentropic flow. At the end the value of the mass flow rate and thrust also calculated.

Calculations:

1. Exit Mach no. correspond to area ratio using isentropic table

   Mach at exit (Me) = 2.875

2. Stagnation Density (Inside chamber)

   $$\rho = \frac{P}{RT} = 6.5853 \text{ Kg/m^3}$$

3. Throat velocity (M=1) at T = chamber stagnation temperature =1200K

   $$V = \sqrt{\frac{2\gamma.R\,T}{\gamma+1}} = 633.8769 \text{ m/s}$$

4. Throat density (using isentropic flow chart)

   Density at throat is = density in chamber*.633938

   = 4.17498Kg/m^3

5. Mass flow rate

   = (throat density* throat velocity* throat area)

   = 4.17498*633.8769*0.00384845

   10.1846 Kg/s

6. Throat pressure using isentropic table for Mach no = 1

   Pt = pressure in chamber* .5282

   = 1197957.6

7. Pressure at exit using isentropic flow relation at Mach No. = 2.88

   Pe = pressure chamber *.03263

   = 74004.84Pa

Note: exit pressure is 74004.84 greater than 39365 Pa hence, system is Under-expanded

8. Exit temperature using isentropic flow table for Mach No. =2.88

   Te = chamber temperature*.3761

   = 451.32K

9. Exit velocity

   Ve = Me*(1.4*287*Te)^2

   = 1226.4214m/s

Now, we have all value to calculate the thrust at the exit of the nozzle and the calculated value is

T= 12991.89144 N

5. Result comparison and discussion

We have performed the simulation of the nozzle, if we take a look in fig. 3 the variation of the Mach number, we can see the flow is expanding outside the nozzle exit that is under-expanded condition, this is can also be verify by the analytical solution, since the exit pressure is greater than outlet pressure of the nozzle. Also, if we see the flow components as shown in fig. 3, we can see boundary layers, slip lines, bow shock, etc. all of these points confirm our result look correct, this a sanity check.

Now, we will compare the result of numerical and analytical result as follow;

1. Mach Number
   Numerical simulation calculation of the average Mach number is 2.69 and analytical calculated value of the Mach Number for an Isentropic flow is 2.88

2. Mass Flow Rate
   Average numerical simulation value of the mass flow rate is 10.3925 Kg/s and analytical calculated value for the mass flow rate is 10.1486 Kg/s

3. Thrust
   Average thrsut value for numerical simulation is 13073.4854 N and analytical calculated value of thurst is 12991.89144 N

   All the values obtained from the nozzle simulation is very close to analyticaly calculated values by assuming isentropic flow through nozzle. hecne, we can confirm our simularion is correct.
   The method of characteristics

Limitations

There are some limitation involved in the analysis as mention below;

1. First we have to acknownledge that, even we have high performing computers to do the simulation or designing the nozzle, the accuracy of the real life suitation cannot be met.
2. Our numerical calculation and as well as analytical solution is based on some assumption which makes our calculatation slightly inaccurate. For example we have different simulation models used in the simulation like k-elipson, k-omega, etc models which will have different result. Also in the case of the analytical solution we have assumed the flow is isentropic for our calculatation but, in actual, there is some turbulence, viscosity, shock waves, etc. which is not considered in the isentropic flow assumption.
3. Also, in the analytical process human error involved which cannot be ignored.

6. Conclusion

Nozzle Design

We have designed a nozzle using the method of the characteristics for the minimum length of the nozzle it has lower quality of flow than the gradual increasing nozzle which used in the supersonic wind tunnel for very high quality flow at the exit. The minimum length is used in space industries because, we want of minimize the weight of the nozzle which result cost of waste in thrsut to lift the weight of the nozzle. We have designed the spersonic CD- nozzle a open source MATLAB code to generate the wall points for a perticular designed Mach number such we have parellel and shock free flow at the exit of the nozzle, which require to improve the performanc and the life of the nozzle. Then we inserted these points in the SOLIDWORKS to generate diverging wall curve. The designed length of the nozzle is 137.04mm exit radius is 67.87mm, inlet radius 35mm (input value) and the maximum wall angle of the nozzle at the downstream is 27.26 degree.

Nozzle simulation

We have performed the numerical simulation of the supersonic CD- nozzle using the ANSYS software a mesh is generated having 45064 nodes and 44484 elements. Then the simulation is performed using the density based solution because the fluid that is air is compressible and k-elipson model is used which is used by most of the researcher in their work. And the simulation is performed, the exit data curve for the exit density, pressure velocity and temperature is generated and then then post processing process is performed in the EXCEL to calculate the exit Mach Number Mass flow rate and Thrust value at the exit and the average values are 2.69, 10.3925Kg/s and 13073.4854 N respectively. We also performed analytical solution by taking assumption the flow is isnetropic and the calculated values of the Mach number is 2.88, mass flow rate is 10.1486Kg/s and thrust is 12991.89144 N. the values from both the methodology is quite similar hence, our result validated.

Referencecs

1. Hussain, M.Satya, S.Nagakalyan, (2019).Design and flow analysis of various convergent – Divergent Nozzles by using Computation fluid dynamics - IJIRAE::International Journal of Innovative Research in Advanced Engineering, Volume VI, 368-371. doi://10.26562/IJIRAE.2019.MYAE10087

2. Goyal S, et. al. "CFD Analysis of Supersonic C-D Nozzle for Optimization of Divergent Angle." *The International Journal of Engineering and Science (IJES)*, 10(03), (2021)

3. Mubarak A.K., Tide P.S., (2018) "Design of a double parabolic supersonic nozzle and performance evaluation by experimental and numerical methods", Aircraft Engineering and Aerospace Technology, https://doi.org/10.1108/AEAT-12-2017-0275

4. Md Akhtar Khan, Sanjay K. Sardiwal, M.V.S.Sharath, D.Harika Chowdary, (2013).Design of a Supersonic Nozzle using Method of Characteristics- International Journal of Engineering Research & Technology (IJERT) ISSN: 2278- IJERTV2IS110026

5. Kumar K Naveen et al 2019 J. Phys.: Conf. Ser. 1355 012012

6. F. Ferdaus, R. Sridhar, G.S. Kumar and S. Sivabalan. 2019. Computational Analysis of Straight Nozzle: Technical Note, Int. J. Vehicle Structures & Systems, 11(2), 193-195. doi:10.4273/ijvss.11.2.15.

7. M.Shais Khan, S.Asim Minhaj, “Numerical Analysis of De Laval Nozzle under Surrounding Zone and Compressed Flow” International Journal for Research in Applied Science & Engineering Technology (IJRASET) ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429

8. Stephen A. Whitmore, “Introduction to the Method of Characteristics and the Minimum Length Nozzle” Mechanical and Aerospace Engineering Department, Utah State University

9. S. M. Yahya, “Fundamentals of Compressible Flow with Air Craft And Rocket Propulsion”, New Age International (P) Limited, Publishers, Fifth Edition, 2016

10. John D. Anderson, Jr. “Modern Compressible Flow”, McGraw-Hill Companies, Inc, Publishers, Third Edition

11. https://github.com/Vinayak-D/MOC-MATLAB

12. https://www.grc.nasa.gov/WWW/k-12/rocket/nozzle.html

13. https://www.isro.gov.in/image-galleries