International Journal of Scientific & Engineering Research, Volume 8, Issue 3, March-2017
ISSN 2229-5518

Optimization In DE-Laval Nozzle Design to Increase Thrust

Parashram V. Patil[1], Akshay A. More[2], Shubham M. Patil[3] and Milind R. Mokal[4]
Under the guidance of prof. Amol Y. Kadam[5]
Department of Mechanical Engineering,
Saraswati College of Engineering,Kharghar,
Dist.-Raigarh, Maharashtra(INDIA).
[1]parashramp50@gmail.com
[2]akshaymore969942@gmail.com
[3]patilshubham216@gmail.com
[4]mokal.milind153@gmail.com
[5]amol.9606@gmail.com

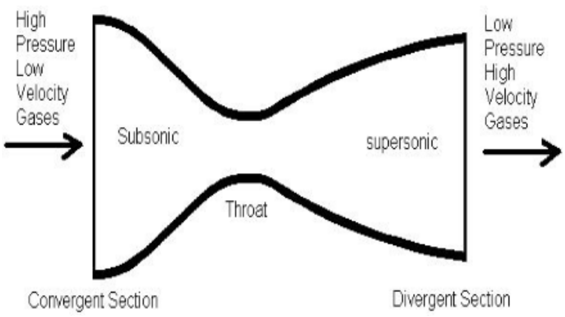
Abstract- Nozzle is mechanical device,used to convert chemical energy of the gases coming out from the combustion chamber into kinetic energy. Also gives direction to the gases coming out from the combustion chamber and converts pressure energy into the kinetic energy. For efficient conversion of pressure energy into kinetic energy convergent-divergent nozzle is being used also known as DE-Laval nozzle. In this paper literature survey has been carried out to increase efficiency of the nozzle by optimizing various performance parameters such as:

1. **Divergent Angle**
2. **Exit Diameter**
3. **Profile of nozzle and**
4. **Cross-sectional area of nozzle**

Key Terms- C-D(convergent-divergent nozzle), CFD(Computational fluid dynamics), ANSYS, FLUENT, DE-Laval Nozzle.

INTRODUCTION

Nozzle is used to carry out for efficient conversion of chemical energy of the gases coming out from the combustion chamber into the kinetic energy. convergent-divergent nozzle is used for efficient conversion of pressure energy into the kinetic energy. The concept of convergent-divergent nozzle was first coined by the Swedish Scientist Gustaf DE-Laval who found that the efficient conversion of the gases is carried out by first narrowing the nozzle, increasing the speed of gases up to the speed of sound and then again expanding to increase the speed of gases beyond the speed of sound. So the velocity of gases at the exit of the nozzle is above the velocity of sound known as supersonic velocity, which is necessary to impart the thrust on the body.



LITERATURE REVIEW

1. Analysis done for different cross section of nozzle a) Rectangular b) Square c) Circular.

From the report G. Satyanarayan et al (2013)[1] it is studied and analyzed that the for the some inlet and boundary conditions the rectangular nozzle gives an increased velocity of 23.93% compared to square nozzle and 24.47% compared to circular nozzle. This is due the pressure as well as the temperature drop into the rectangular nozzle s greater than the other two nozzle. The rectangular nozzle gives an increased pressure and temperature drop of abut 22.93% and 42.56% compared to the square nozzle. Also rectangular nozzle gives increased temperature and pressure drop of about 23.97% and 43.68% compared to the circular nozzle and hence the rectangular cross section nozzle s the best cross section compared to circular and square for same input and boundary condition.

2. *Modeling in the conical nozzle*

From the report Venkatesh V. et al (2015) [2]. it is observed that the result obtained by using fluent After the simulating the transient gas flow by coupled explicit solver it gives 2-D result which show that contour nozzle gives a greater mach number at ext compared to conical nozzle. As the contour nozzle gives maximum expansion ratio compared to the conical nozzle. Hence the conical nozzle has has to be used at sea level where as contour nozzle to be used at higher altitude since greater expansion ratio is required at higher altitude for given length of nozzle. But contour nozzle has complex geometry that is the profile may be parabolic or cubic therefore it is difficult to fabricate.

3. Increasing the thrust by varying the exit diameter of nozzle

From the report Arjun Kundu et al (2016) [3] it is observed that the result obtained after the CFD analysis shows that smaller exit diameter gives greater mach number compared to the larger diameter for the same inlet and boundary condition. The first nozzle with exit diameter 60 cm increase in velocity from inlet to outlet is 137.37 m/s but in second nozzle with exit diameter 53 cm the increase in velocity from inlet to exit is 152.6 m/s. This represents that velocity at exit increases with decrease in diameter. The nozzle with exit diameter 53 cm gives maximum mach number at exit compared to the nozzle with 60 cm diameter.

4. *Flow analysis for various divergent angle*

From the Report of Natta Pardhasardhi et al (2015) [4] Work is done to check the variation in mach number with respect to divergent angle.

From above results it is found that divergence angle is minimum for 20 degrees of divergence angle and further increases with increase in divergent angle.

Divergence Angle(Degree)	Mach number
7	2.917
20	2.84
30	3.06
40	3.19

Table no.01 divergence angle variation[4]

From the report of Karna S. Patel (2014) [5] This paper provides discussion on the study of variation of mach number at the exit of nozzle with variation in the divergent angle with the help of CFD analysis and the objective of the research is to investigate the best suited divergent angle. As well as the phenomenon of oblique shock is visualized to find the suitable divergent angle to prevent shock. Input boundary conditions In this paper it was found that with increase in divergence angle from 5 degree to 15 degree oblique shock waves keeps on reducing and was completely eliminated at 15 degrees of divergence angle.

From the report of Nikhil D. Deshpande et al (2014) [6] and Bogdan Alexandru Belega et al (2015) [11]The theoretical and CFD analysis of De-Laval nozzle have been carried out which was aiming to provide theoretical formulae to calculate velocity, pressure, temperature at every section of nozzle. the validation of these is carried out using CFD software ANSYS FLUENT. Results obtained theoretically are verified with the results obtained by CFD and it was found that results are identical. There are some variation in the results, because theoretical study does not account boundary layer effect, shock wave, radial velocity component, wall friction and so on. But the the variation is quite insignificant .

From the report of Mohan Kumar G et al (2013) [10] design and optimization of De-Laval nozzle had been carried out to prevent shock induced flow separation. The aim was to achieve maximum thrust without flow separation due to flow separation. For maximum thrust and efficiency, the direction of flow of stream through nozzle should be axial. Flow conditions were selected based on pressure, temperature and gases that are available at the exit of the combustion chamber. To prevent shock induced flow separation, overexpansion conditions and optimum expansion conditions are studied. And it was found that optimum expansion is critical for efficient operation of nozzle

From the report of Sourabh Shriwas et al (2015) [13] reduction of jet noise in exhaust nozzle of aircraft engine have been studied. It was fond that various methods could be adopted for noise reduction. Such as use of Chevron nozzle which consist of sharp saw tooth at the aft which causes smooth mixing of hot steam with the bypass steam hence turbulence is reduced. Also making use of nano materials with higher density are capable of absorbing noise, increasing bypass ratio of turbofan, acoustic liners can also be implemented for noise reduction.

From the report of Vignesh. Met al *(2013)* [8] conceptual Design of Short Take-Off Supersonic Aircraft with cold flow Nozzle is carried out in 2013. The rectangular C-D nozzle is altered type DE-Laval nozzle having reduced frontal area. The ultimate idea is to reduce the takeoff distance. By placing the C-D nozzle under the wing of Military aircraft,flow can be accelerated at the choked flow condition resulting in sudden increase in thrust and short takeoff. Numerical study were carried out which highlighten that for the design,Shock Waves are not formed in the divergent section and also takeoff distance reduces by 50%.

FUTURE SCOPE

1. To increase the thrust in passenger drone.

2. To increase the thrust imparted in rocket nozzles. So that propellant requirement is to be reduced.

LIMITATION

1. The design of parabolic or cubic profile contour of nozzle is complex which would results into increased manufacturing cost.

2. New integrated structure of nozzles could possibly decrease the mass flow rate.

3. The material required should have low thermal expansion coefficient and should posses high strength.

CONCLUSION

The flow through C-D nozzle had been studied through various literature review. It was found that for smaller exit diameter more thrust could be achieved[8]. For smaller diameters there are low chances of flow separation hence thrust exerted on the body is larger in case of C-D nozzle with smaller diameter than the larger diameter. Hence introducing a set of nozzle and allowing flow to pass through it can possibly give more thrust than the single nozzle.

REFERENCES

[1] G. SATYANARAYANA, 2. Ch. VARUN, 3. S.S. NAIDU, “CFD analysis C-D nozzle”, ACTA TECHNICA CORVINIENSIS- Bulletin of engineering, Tome VI, ISSN 2067-3809, 2013.

[2] Venkatesh .V, C Jaya pal Reddy, “Modeling and Simulation of Supersonic nozzle using Computational Fluid Dynamics”, International Journal of Novel Research in Interdisciplinary Studies, Vol. 2, Issue 6, pp: (16-27), 2015.

[3] Arjun Kundu, Devyanshu Prasad, Sarfraj Ahmed, “Effect of Exit Diameter on the performance of Converging-Diverging Annular Nozzle using CFD”, International Journal of

Innoavatine Research in Science, Engineering and technology, Vol. 5, Issue 6, June 2016.

[4] Pardhasaradhi Natta,V.Ranjith Kumar, Dr. Y. V. Hanumantha rao, "Flow Analysis of Rocket nozzle using Computational Fluid Dynamics (CFD)", Engineering Research and Applications (IJERA) ISSN: 2248-9622 www.ijera.com Vol. 2, Issue 5, September- October 2012, pp.1226-1235.

[5] Karna S. Patel, "Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD)", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. IV (Nov-Dec. 2014), PP 01-10.

[6] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap, "Theoretical and CFD analysis of De Laval Nozzle", International Journal of Mechanical And Production Engineering, ISSN: 2320-2092, Volume- 2, Issue- 4, April-2014.

[7] Sourabh Shriwas, Sachin Shah, Prashant Shah, Kalpit P. Kaurase, "Reduction Of Jet Noise In the Aircraft Nozzle", International Journal Of Research In Aeronautical And Mechanical Engineering ISSN (ONLINE): 2321-3051 Vol.3 Issue.3, March 2015. Pgs: 68-74.

[8] Vignesh. M , Sathish. k , Adrin Issai Arasu. L ,Dharun Lingam. K , and Sanal Kumar. V. R, "Conceptual Design of Short Takeoff Supersonic Aircraft with Cold Flow Nozzles", 3rd International Conference on Trends in Mechanical and Industrial Engineering (ICTMIE'2013) , Kuala Lumpur (Malaysia), January 8-9, 2013.

[9] K. M. Pandey, A. P. Singh, "CFD analysis Of Conical Nozzle For MACH 3 at Various Angles of Divergence with Fluent Software", International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, ISSN: 2010-0221.

[10] Mohan Kumar G , Dominic Xavier Fernando, R. Muthu Kumar, "Design and Optimization of De Laval Nozzle to prevent shock induced Flow Separation", Advances in Aerospace Science and Applications, ISSN 2277-3223 Volume 3, Number 2 (2013), pp. 119-124.

[11] Bogdan Alexandru Belega, Trung Duc nguyen, "Analysis of flow in Convergent-Divergent Rocket Engine Nozzle using Computational Fluid Dynamics", International Conference of Scientific Paper AFASES 2015 Brasov, 28-30 May 2015.

[12] M. C. Louwerse, H.V. Jansen, M.N.W.Groenendijk, "Nozzle Fabrication For Micro Propulsion Of a Microsatellite".

[13] Sourabh Shriwas, Sachin Shah, Prashant Shah, Kalpit P. Kaurase, "Reduction Of Jet Noise In the Aircraft Nozzle", International Journal Of Research In Aeronautical And Mechanical Engineering ISSN (ONLINE): 2321-3051 Vol.3 Issue.3, March 2015.

[14] Lars Davidson, "An Introduction to turbulance model", Department of thermo and fluid dynamics, Chalmers university of technology, Goterberg, Sweden, November, 2013.

[14] Karna s. Patel, "CFD analysis of an aerofoil", International Journal of engineering research.

[15] David R. Greatrix, "Regression rate estimation for standard-flow hybrid rocket engines", Aerospace Science and Technology 13, 358-363, 2009.

[16] Manna, D Chakraborty ―Numerical Simulation of Transverse H2 Combustion in Supersonic Airstream in a Constant Area Duct‖, Vol. 86, November, 2005.