International Journal of Mechanical Engineering (IJME)
ISSN(P): 2319-2240; ISSN(E): 2319-2259
Vol. 8, Issue 4, Jun - Jul 2019; 1-14

International Academy of Science, Engineering and Technology
Connecting Researchers; Nurturing Innovations

EFFECT OF CD NOZZLE AT VARIOUS NOZZLE PRESSURE RATIOS WITH VARYING DEFLECTION ANGLES

Bhagyalakshmi A[1], Dharini S[2], Sheela M[3] & Manikandan S[4]

[1,2,3]Research Scholar, Department of Aeronautical Engineering, Jeppiaar Engineering College, Chennai, Tamil Nadu, India

[4]Assistant Professor, Department of Aeronautical Engineering, Jeppiaar Engineering College, Chennai, Tamil Nadu, India

ABSTRACT

The thrust vector control is the ability to control the direction of thrust in a vehicle and it also plays a vital role in controlling the altitude and the angular velocity. The aim of this paper is to increase the mass flow rate and thrust vectoring of the nozzle by deflecting the wall at the divergent section of the nozzle at various deflection angles. A contour C-D nozzle is designed with a domain using Hypermesh and analyzed using ANSYS Fluent. The domain is created to capture the jet plume outside the nozzle. Various flow parameters such as pressure, density, Mach number are noted for various nozzle pressure ratios with respect to various deflection angles. The deflections are made at the divergent section of the nozzle and the analysis is carried out. The results of the base model and the other deflected models are compared to obtain the better nozzle geometry for thrust vectoring.

KEYWORDS: *Nozzle, Deflection, Thrust Vectoring, Efficiency*

Article History

Received: 15 May 2019 | Revised: 27 May 2019 | Accepted: 13 Jun 2019

INTRODUCTION

A nozzle is a simple device, a specially shaped tube through which hot gases flow and expand to a higher velocity. A nozzle can be a convergent nozzle or a combination of convergent and divergent section. A typical fixed convergent-divergent nozzle is used in ramjets and rockets in order to achieve high Mach number. In a CD nozzle, the hot exhaust from the combustion chamber which is expanded by the turbine enters the nozzle and converges down to the throat of the nozzle. The throat of the nozzle is designed to choke the flow and set the mass flow rate through the system. The flow in the throat is increased to sonic which means the Mach number is equal to one in the throat. Downstream of the throat, the nozzle diverges and the flow is entropically expanded to a supersonic Mach number that depends on the area ratio of the exit to the throat of the nozzle. The expansion of supersonic flow in the divergent section causes the static pressure and temperature to decrease from the throat to the exit. Hence, the amount of expansion also determines the exit pressure and temperature.

The Bell-shaped or contour nozzle is the most commonly used shaped rocket engine nozzle. It has a high angle expansion section behind the nozzle throat. The expansion section is followed by a gradual reversal of nozzle contour slope and as a result of that at the nozzle exit, the divergence angle is small, usually less than a 10 degree half angle.

Thrust vectoring also thrust vector control or TVC, is the ability of an aircraft, rocket, or another vehicle to change the direction of the thrust from its engine(s) or motor(s)so that the control of the attitude or angular velocity of the vehicle can be obtained. The thrust vectoring is the primary means of attitude control in rockets and ballistic missiles.Thrust-Vectoring flight control (TVFC) is achieved by deflecting of the aircraft jets in a certain angle so that the direction of thrust can be varied. The deflection of the jets in various directions creates desired forces and moments which enables complete directional control of the aircraft flight path.

Implementation of TVFC can be done in a variety of nozzles by applying both mechanical and fluidic. The example of this condition includes convergent and convergent-divergent nozzles that may be fixed or geometrically variable. It also includes external thrust vectoring control mechanisms within a fixed nozzle, such as rotating cascades and rotating exit vanes. The geometry of the nozzle itself may vary from two-dimensional (2-D) to axisymmetric or elliptic. To achieve TVFC, the number of nozzles used can vary from one on a CTOL aircraft and in case of a STOVL aircraft, a minimum of four can be required.

As the paper is focused about the concept of thrust vectoring in the nozzle without using external devices, the analysis of contour nozzle is carried out by deflecting it at the divergent section by certain angle and the nozzle with maximum thrust vectoring ability is considered.

METHODOLOGY

Understanding the Flow Physics over the convergent-divergent nozzle and the effect of deflection angle at various nozzle pressure ratio using available research and technical papers, CAD Modeling of a convergent-divergent nozzle model without deflection and with deflection are designed. The variation in flow properties due to the wall deflection at the divergent section is observed. The performance of the nozzle at various nozzle pressure ratio is analyzed and a detailed study of the performance of convergent-divergent nozzle was carried out.

Geometry

The contour section of the nozzle is designed using the concept of the method of characteristics. The Method of Characteristics is a numerical method for solving non-linear equations of motion for in-viscid, irrotational flows, from which the coordinates of the contour section can be obtained. Other CFD methods such as finite element method can be used for the designing the nozzle profile but these methods require more extensive numerical calculations. The Method of Characteristics is based on the isentropic flow equations. Using the method of characteristics, the nozzle geometry is set with corresponding reference parameters from the reference papers.

Design

In order to achieve best results with a fine mesh, the models were designed in HYPERMESH Design modeler. The meshing operations were performed in HYPEMESH to achieve a finer mesh that results inaccurate results. The next step involves the numerical simulation of the meshed models. The meshed models were computationally analyzed in ANSYS FLUENT 16.0 using the steady two-dimensional density based energy equation. The turbulence characteristics were simulated using the k-□ turbulence model with standard wall functions. The contours of static pressure, density, Mach number, turbulence kinetic energy were taken along the model length for investigating the flow performance with respect to the changing angle at the divergent section of the nozzle and the effect relating thrust vectoring of the nozzle. Finally, the graphs were plotted for the above-mentioned parameters along the center-line of the model length so as to compare the performance of the models chosen.

CAD MODEL OF NOZZLE

The cad model of the nozzle is designed using the Solid works tool which will provide accurate results since the mesh generated is very fine for analysis that gives more accuracy. The geometry of the model is set by using the method of characteristic for the Mach number and the obtained coordinate points are used to set the dimension for the nozzle.

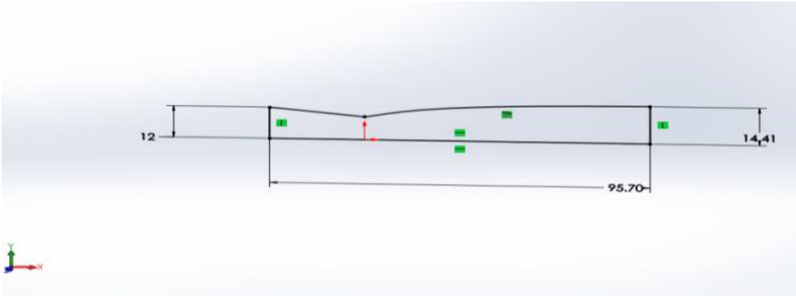


Figure 1: CAD Model of Base Contour Nozzle

Figure 2: Base Contour Nozzle with Domain

Figure 1 and Figure 2 shows the modelling of the base contour nozzle. Similarly, the nozzle with deflections is designed. The deflection angles chosen are 5.04 degree, 10.06 degree which is positioned in the divergent section at a distance 66% from the throat portion of the nozzle. Further, two more models are designed with a double deflection on the same side of the divergent section of the nozzle in which the first angle is taken as mentioned in the above cases. The second angle is taken as 2.8 degrees for the deflection and this is positioned at 76% of the distance from the nozzle.

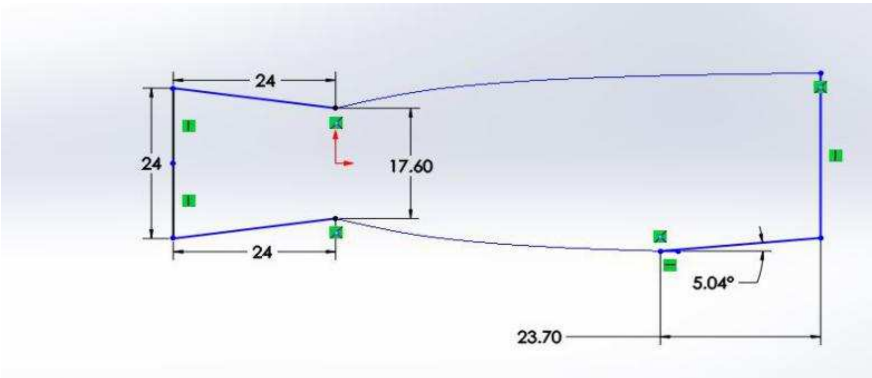


Figure 3: Contour Nozzle with 5.04 -Degree Deflection

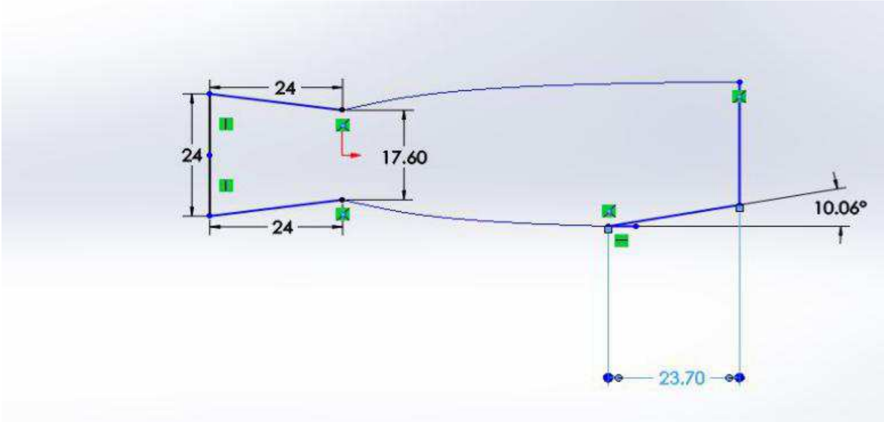


Figure 4: Contour Nozzle with 10.06 -Degree Deflection

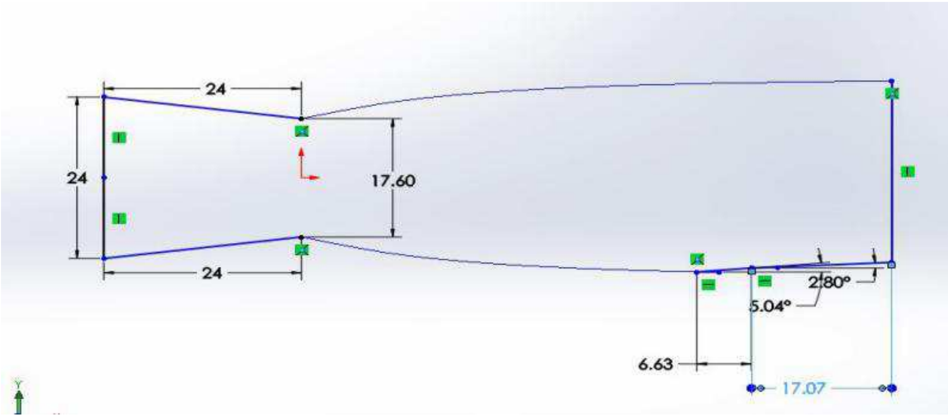


Figure 5: Contour Nozzle with 5.04 - 2.8 -Degree Deflection

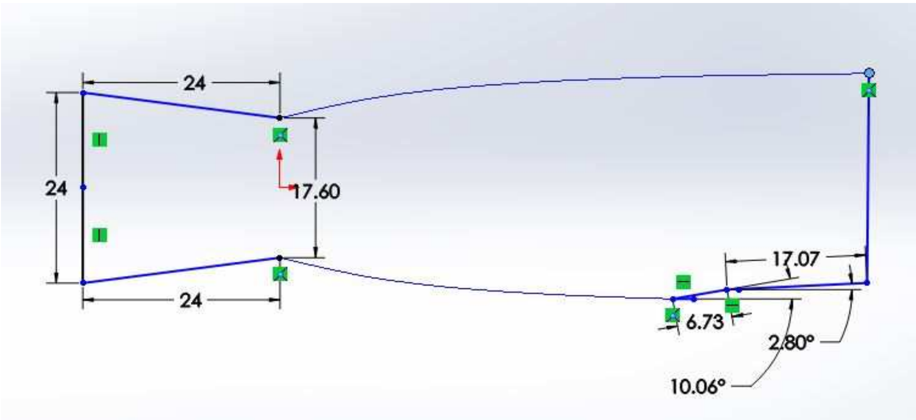


Figure 6: Contour Nozzle with 10.06 - 2.8 -Degree Deflection

Figure 6 Shows the Cad Modeling of Various Deflected Contour Nozzle Models Considered for the Analysis.

MESH OF THE MODEL

Meshing for the designed geometry of the contoured nozzle is applied with hyper mesh tool. The mesh refinement is taken fine to obtain better and accurate results for the configurations. The mesh size is taken as 0.1mm in the nozzle and 1.5 at the nozzle domain region. The variation in the mesh size is to reduce the complexity of the mesh during analysis and for time consumption.

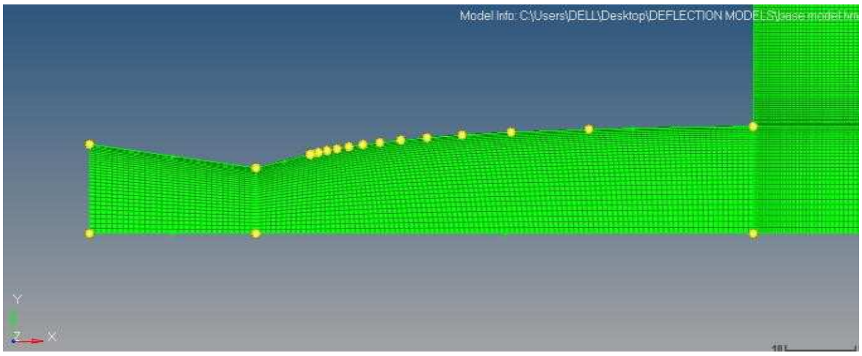
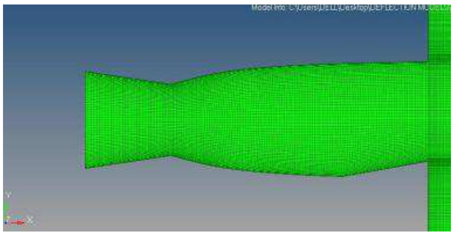


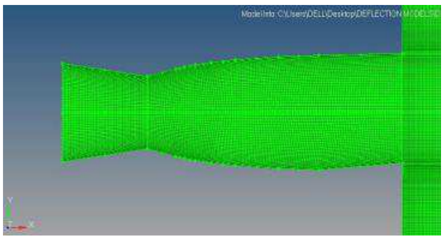
Figure 7: Mesh Model of Base Contour Nozzle

Figure7 Shows the Mesh Model of the Base Contour Nozzle Which Is Created In Hypermesh Tool. Symmetrical meshed base model The mesh refinement is done with biasing which concentrates the mesh at the regions of the nozzle where the results to be captured needs a higher extent of accuracy. Figure.8 shows the mesh refinement of the nozzle designed.

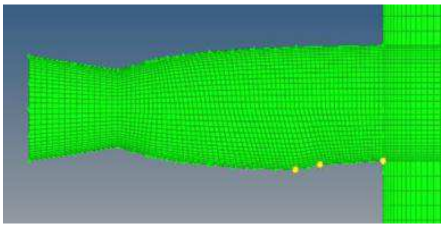
Figure 8: Mesh Refinement of the Nozzle

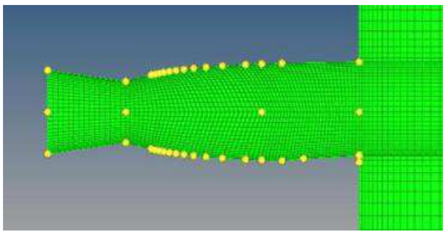
Figure 9 (A) to (D) Shows the Meshed Models of Contour Nozzle with Deflections at the Divergent Section





(A) **(B)**





(C) **(D)**

Figure 9: Meshed Models - (A) 5.04 Degree Model (B) 10.06 Degree Model(C) 5.04-2.8 Degree Model (D) 10.06-2.8 Degree Model

BOUNDARY CONDITIONS

The base model and the various configurations are applied with different boundary conditions to obtain a result with a comparison based on the variation of the boundary condition. The inlet pressure is given as 5 bar. The initial gauge pressure is given as 4 bar. The nozzle pressure ratio is set to be 5 for the base model configuration. The nozzle area ratio changes with changing deflection angle. The results of the various configuration vary based on the boundary condition appliedto each model. The second, third and the fourth configuration are given fixed nozzle pressure ratio. The fourth and fifth configuration with double deflection is also given the same fixed nozzle pressure ratio and the results obtained are considered for comparison of the Mach disk decay and the flow deflection. The same set of configurations are applied with 7 NPR and results are compared.

RESULTS AND CONCLUSIONS

The base contour CD nozzle is designed and analyzed in the ANSYS Fluent with the pressure inlet conditions of 5 NPR and 7 NPR respectively. Similarly, the deflected configurations are also designed and analyzed at the same boundary conditions. The density-based computational analysis is carried out in the various models since the analysis is carried out for the compressible flow. After the convergence of the solution, the post-processing of the results is carried out from which the various contours and the plots can be obtained for each configuration. The results obtained are compared with the results obtained. From the results, the variation of Mach number, the change in direction of the plume is noted. As this project is focused on thrust vectoring, the nozzle configuration with better deflection of the flow without much variation in the exit Mach number is considered.

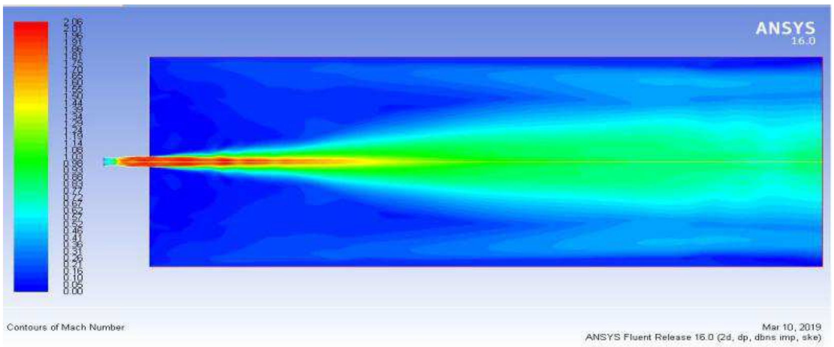


Figure 10: Mach Contour of the Base Model at 5 NPR

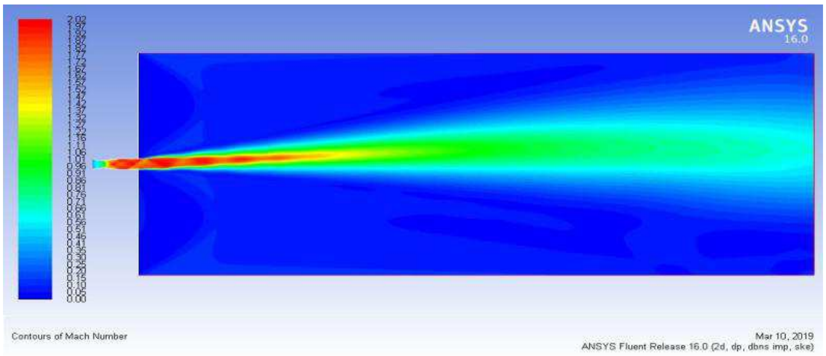


Figure 11: Mach Contour of 5.04 Degree Deflected Model at 5 NPR

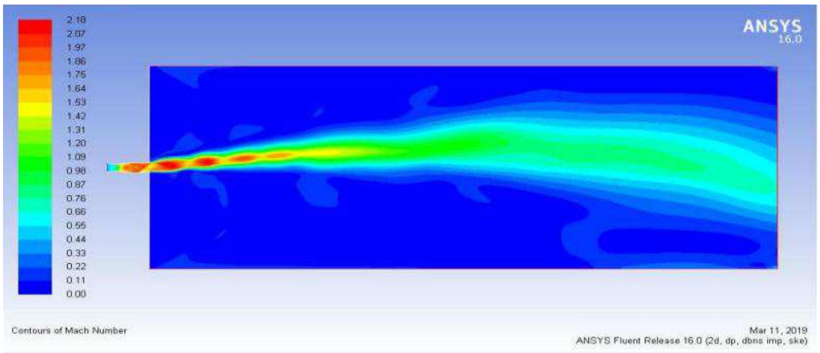


Figure 12: Mach Contour of 10.06 Degree Deflected Model at 5 NPR

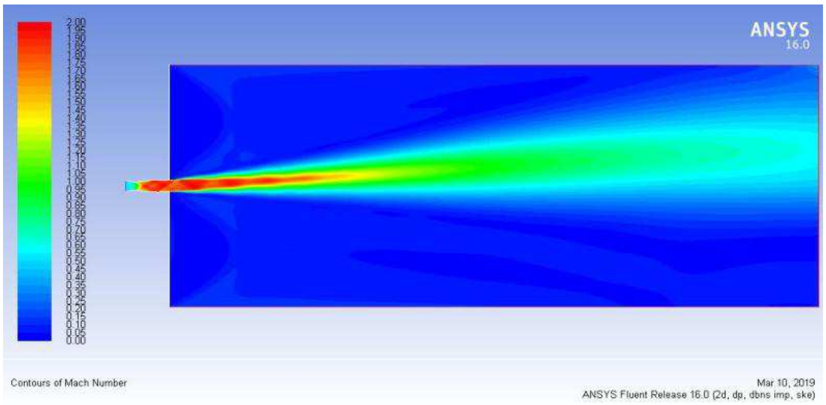


Figure 13: Mach Contour of 5.04-2.8 Degree Deflected Model at 5 NPR

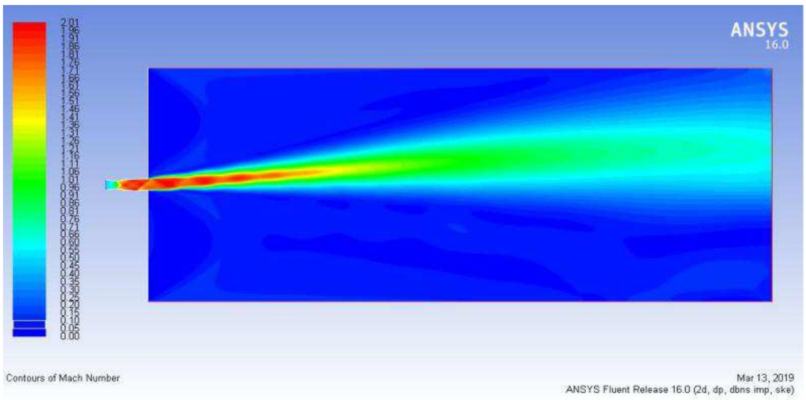


Figure 14: Mach Contour of 10.06-2.8 Degree Deflected Model at 5 NPR

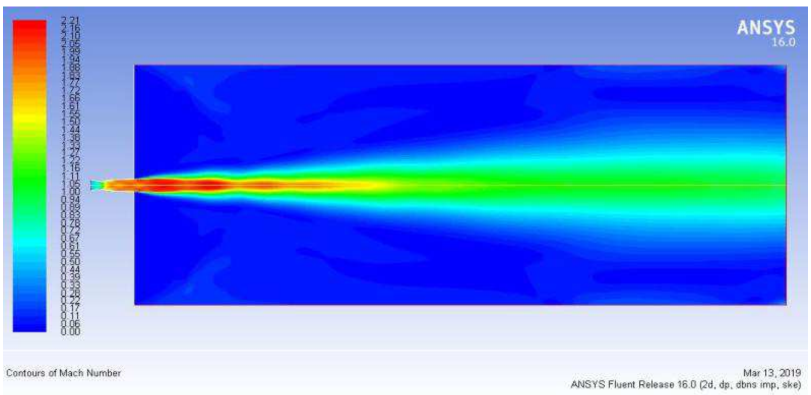


Figure 15: Mach Contour of the Base Model at 7 NPR

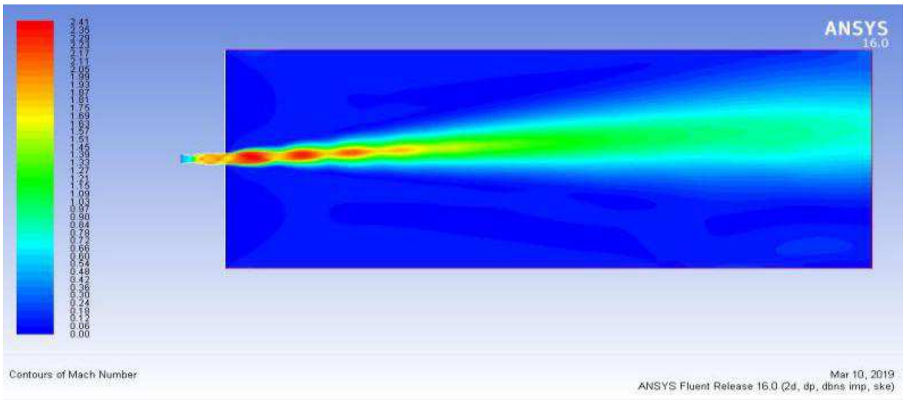


Figure 16: Mach Contour of 5.04 Degree Deflected Model at 7 NPR

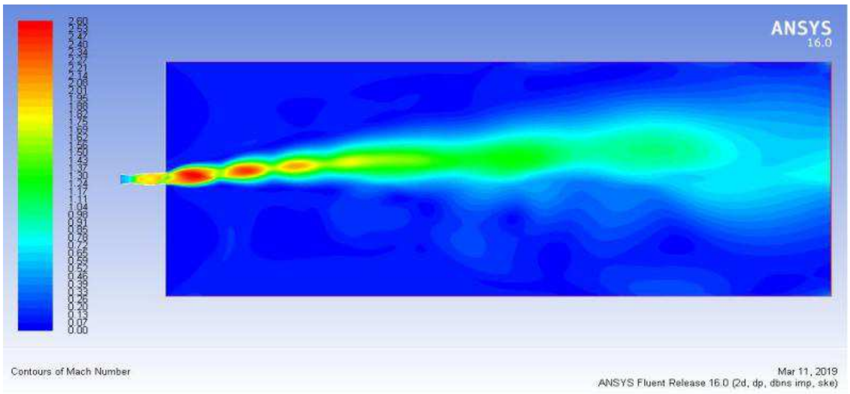


Figure 17: Mach Contour of 10.06 Degree Deflected Model at 7 NPR

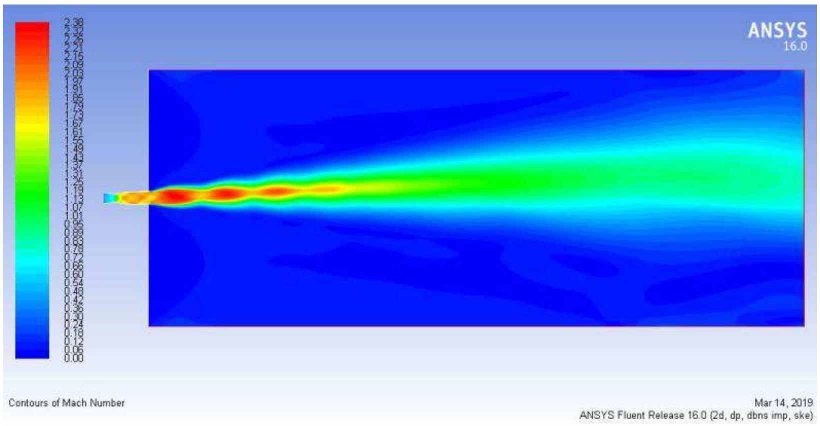


Figure 18: Mach Contour of 5.04-2.8 Degree Deflected Model at 7 NPR

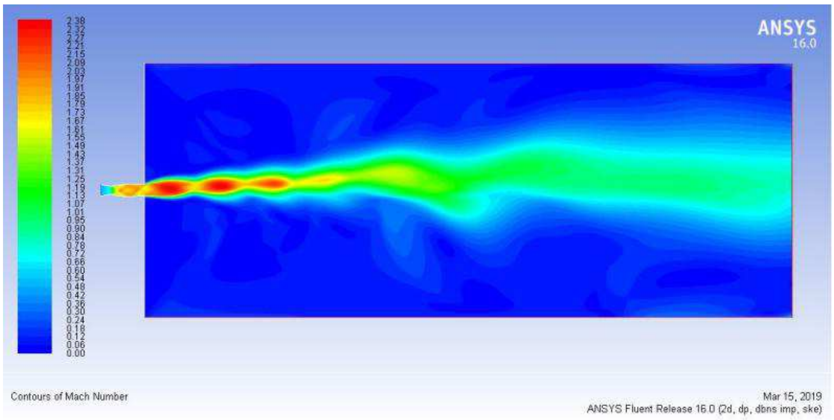


Figure 19: Mach Contour of 10.06-2.8 Degree Deflected Model at 7 NPR

From the above contours, the variation in Mach number and the formation of the jet plume can be observed. Figure.10 shows the mach contour of the base model at 5 NPR. From the mach contour, it is observed that the Mach number increases in the divergent portion as the flow expands after the throat portion. Figure.11 shows the mach contour of the deflected model with 5.04-degree deflection at the divergent portion of the nozzle exit and it can be observed that when a deflection is made in the nozzle, the exit Mach number is reduced and the flow has been deflected to certain angle with respect to the center line of the nozzle. Figure.12 shows the Mach contour of 10.06 degree deflected model under the condition of 5 NPR. Here, the deflection of the flow can be observed very clearly but the Mach number is increased compared to the base model. As discussed above, the deflection at the divergent portion causes the Mach number to be reduced than on the base model. Figure.13 shows the mach contour of 5.04-2.8 double deflected model at 5 NPR. As a result of the introduction of double deflection near the nozzle, the Mach number has been reduced compared to the base model. It can be observed that the flow deflection angle is reduced than the angle deflected in the 5.04 degree deflected model. Figure.14 shows the mach contour of 10.06-2.8 double deflected model at 5 NPR. The Mach number is increased as the NPR is increased. It is observed that the flow deflection angle is reduced than observed in the 10.06-degree single deflection.

The base model is analyzed at the pressure inlet of 7 NPR which can be observed from Figure.15. It can be observed that as the NPR is increased, the Mach number is also increased. Figure.16 shows the mach contour of deflected model analyzed at 7 NPR. As observed in the previous case, the Mach number has been increased considerably compared to the model analyzed under the condition of 5 NPR. Figure.17 shows the mach contour of 10.06 degree deflected model at 7 NPR. As a result of increasing the nozzle pressure ratio, the Mach number is increased. From the contour, it can be observed that when the nozzle pressure ratio has been increased from 5 to 7, the jet plume diameter has been increased. Figure.18 shows the mach contour of 5.04-2.8 double deflected model at 7 NPR. As observed in the previous cases, increasing the NPR increases the mach number and also the jet plume diameter is increased. Figure.19 shows the mach contour of 10.06-2.8 double deflected model at 7 NPR. As the NPR is increased, the Mach number is also increased and the diameter of the jet plume is also increased. The flow deflection angle is reduced as the NPR is increased.

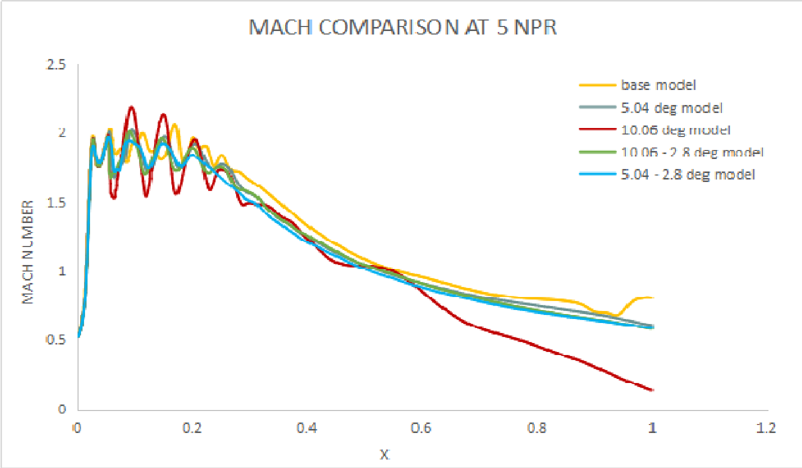


Figure 20: Mach Variation Plot at 5 NPR

Figure.20 shows the comparison of Mach number of various configurations at the pressure inlet condition of 5 NPR. Inside the nozzle, the flow properties are almost the same in all the configurations. It can be observed that there is a difference in the decaying behavior of the jet plume between the base model and the deflected models. As the angle of deflection increases, the period of decay decreases. From the graph, it can be observed that in the model with 10.06-degree deflection, the plume decays rapidly when compared to other configurations. In the same model, the Mach number of Mach disks formed is also increased.

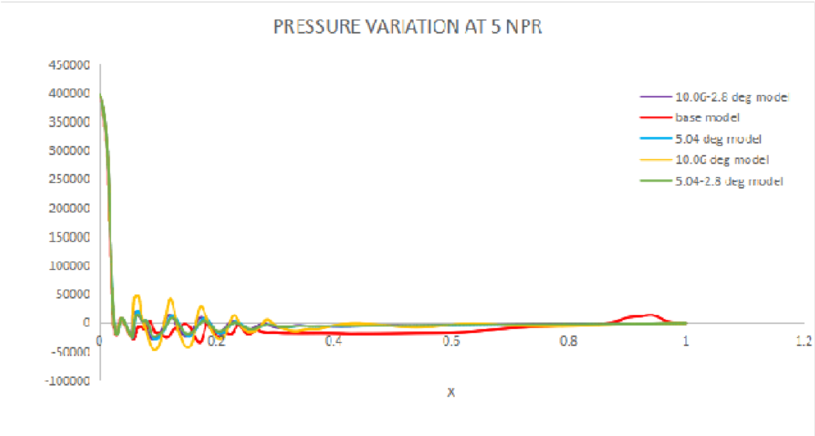


Figure 21: Pressure Variation Plot at 5 NPR

Figure 21 shows the comparison of pressure variation of various configurations at 5 NPR. The rapid variation in the pressure indicates the presence of Mach disks. From the graph, it is observed that along with the domain, the pressure gradually and when it reaches the operating condition, the jet plume decays and becomes subsonic as it moves away from the nozzle exit. It is clear from the graph that for 10.06 degree deflected model, the Mach disks formed are of high strength where there exists a larger pressure variation among all the other configurations.

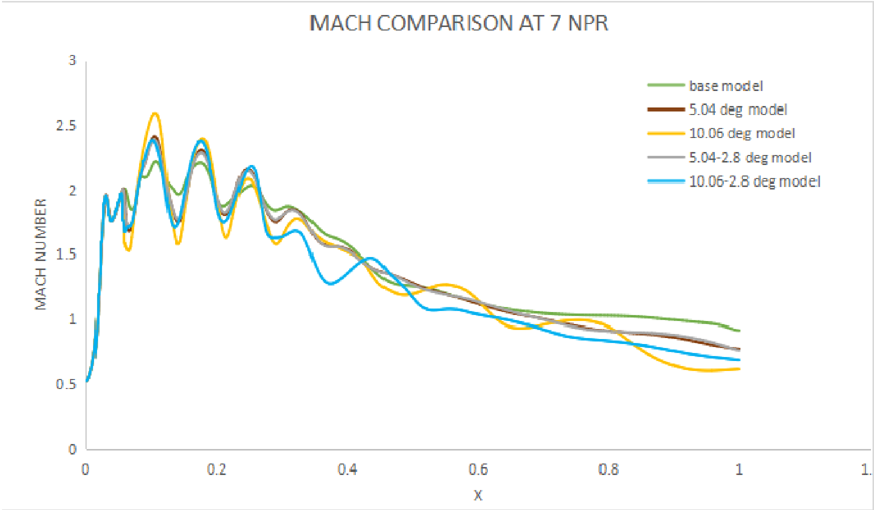


Figure 22: Mach Variation Plot at 7 NPR

Figure 22 shows the comparison of Mach number of various configurations at the pressure inlet condition of 7NPR. The change in flow properties is similar to those observed in the previous case of 5 NPR. But the Mach number is increased than observed in the previous case of 5 NPR.

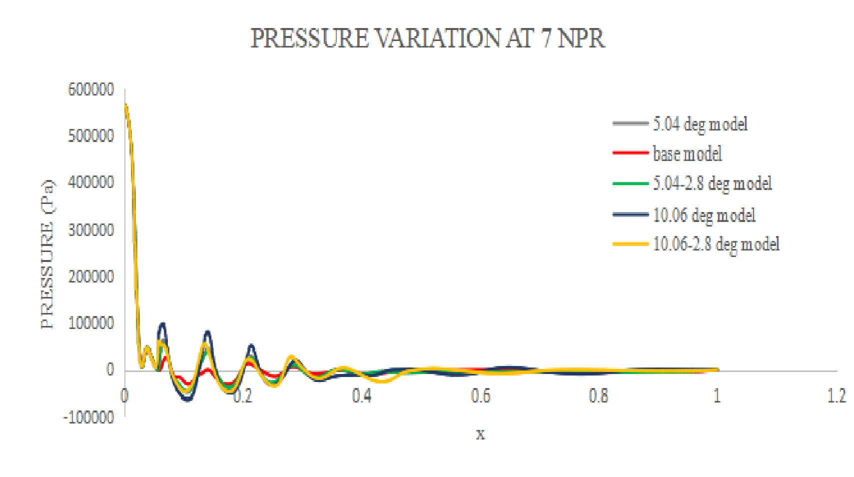


Figure 23: Pressure Variation Plot at 7 NPR

Figure 23**:** shows the comparison of pressure variation of various configurations at 7 NPR. The observations are similar to the previous case of 5 NPR. Also, it is found that the pressure variations are larger at 7 NPR than at the condition of 5 NPR.

Table 1: Comparison of Maximum Mach Number

Configurations	Maximum Mach Number	
	AT 5 NPR	AT 7 NPR
Base model	2.06	2.21
5.04 degree model	2.02	2.41
10.06 degree model	2.18	2.60
5.04-2.8degree model	2	2.38
10.06-2.8degree model	2.01	2.38

From the above table, it can be noted that the maximum Mach number of various models at 5 and 7 NPR respectively. At 5 NPR, When deflection is made in the divergent portion, the Mach number is initially decreased for 5.04-degree deflection, then the mach number increases for 10.06-degree deflection and for the double deflected model, the mach number decreases. It can be observed that when NPR is increased from 5 to 7, the Mach number is increased.

Table 2: Comparison of Flow Deflection Angle

Configurations	Flowdeflection Angle(Degree)	
	AT 5 NPR	AT 7 NPR
Base model	-	-
5.04 degree model	3.8	3.24
10.06 degreemodel	6.49	5.95
5.04-2.8degree model	2.32	2.69
10.06-2.8degreemodel	4.68	3.44

From the above table, the variation in flow angle for various configurations is compared at 5 NPR and 7NPR respectively. In the single deflected model, when the deflection angle is increased, the flow deflection is also increased. Inthe double deflected model, the flow deflection is reduced when compared to the single deflected model. It is observed that when NPR is increased, the flow deflection angle is reduced.

The analysis of the base contour nozzle and the other deflected models are carried out under two different nozzle pressure ratios. It is found that the configurations with a single deflection at one side of the divergent portion are better than the configurations with a double deflection on the same side of the divergent portion.

By analyzing the overall results, it is observed that the model with 10.06-degree deflection serves to be the better nozzle geometry for the thrust vectoring with minimal loss in thrust.

REFERENCES

1. *"Numerical Investigation of Supersonic Nozzle Flow Separation", Q. Xiao and H. M. Tsai, National University of Singapore, Singapore 119260, Republic of Singapore and D. Papamoschou,University of California, Irvine,Irvine, California 92697-3975*

2. *E.M.S Ekanayake, J.A.Gear and Y.Ding, (2010). "Numerical simulation of a supersonic convergent divergent nozzle with divergent angle variation for under expanded condition", Auckland, newzealand, December 2010.*

3. *Lokeswari k, Shanmuga priya M, Shobiya rani T, S.Manikandan(2015). "Numerical analysis od supersonic c-d nozzle with varying exit deflection angles".*

4. *B.V.Naga sudhakar, B purna Chandra sekhar, P narendra mohan, MD Touseef (2016). "Modeling and Simulation of Convergent Divergent Nozzle using Computational Fluid Dynamics", published in International Research Journal of Engineering and Technology volume.3, Issue:08, Aug2016.*

5. *Gutti Rajeswara Rao, U.S.Ramakanth, A.Lakshman, (2013). "Flow analysis in a convergent-divergent nozzle using CFD", published in International Journal of Research in Mechanical Engineering (IASTER), vol.1, issue 2, pp 136- 144, Oct- Dec, 2013. ISSN 2345-5188.*

6. *Wang, X., Heberlein, J., Pfender, E., & Gerberich, W. (1999). Effect of nozzle configuration, gas pressure, and gas type on coating properties in wire arc spray. Journal of Thermal Spray Technology, 8(4), 565-575.*

7. *Chauvet, N., Deck, S., and Jacquin, L., "Shock patterns in a slightly underexpanded sonic jet controlled by radial injections," Physics of Fluids, Vol. 19, 2007 Article No. 048104.*

8. *"Scott""Shock Diamond sandMach Disks" ,Aerospace WebURL: http://aerospaceweb.org/question/propulsion/q0224.shtml[cited 17 March 2008].*

9. *"Romine, G. L., "Nozzle Flow Separation," AIAA Journal, Vol. 36,No. 9, 1998, pp. 1618–1625.*

10. *Hunter, C. A., "Experimental, Theoretical, and Computational Investigation of Separated Nozzle Flows," AIAA Paper 98-3107,1998.*

11. *Carlson, J. R., "A Nozzle Internal Performance Prediction Method," NASA TP 3221, 1992.*

12. *"Analysis of Mach Disks from an Underexpanded Nozzle using Experimental and Computational Methods",Alexander C. Bayeh, Department of Aerospace Engineering, Texas A&M University, College Station, TX, 77*

13. *Papamoschou, D., and Zill, A., "Fundamental Investigation of Supersonic Nozzle Flow Separation," AIAA Paper 2004-1111,Jan. 2004.*

14. *Bourgoing, A., and Reijasse, P., "Experimental Analysis of Unsteady flows in a Supersonic Planar Nozzle," International Symposium on shock Waves, Vol. 14, No. 4, Springer, New York, 2001, pp. 251–258.*

15. *Deck, S., Hollard, R., and Guillen, Ph., "Numerical Simulation of steady and Unsteady Separated Nozzle Flows," AIAA 2002-0406, 2002.*

16. *Sadeghi, M., Yang, S., Liu, F., and Tsai, H. M., "Parallel Computation of Wing Flutter with a Coupled Navier–Stokes and CSD Method,"AIAA Paper 2003-1347, 2003.*

17. *Hagemann, G., Frey, M., and Koschel, W., "Appearance of Restricted Shock Separation in Rocket Nozzles," Journal of Propulsion and Power, Vol. 18, No. 3, 2002, pp. 577–584.*