Numerical Analysis of Supersonic C-D Nozzle with Varying Exit Deflection angles

Lokeswari K[1], Shanmuga Priya M[2], Shobiya Rani T[3], S.Manikandan[4]

[1,2,3]U.G Scholar, Department ofAeronautical Engineering, Jeppiaar Engineering college, Chennai, India.
[4]Assistant professor, Department of Aeronautical Engineering, Jeppiaar Engineering College, Chennai, India.

---***---

Abstract- *Understanding the flow physics behind this jet control is always tedious as the expansion process through the divergent portion cannot be predicted accurately. This paper focuses on numerical investigation on the flow physics at the divergent section with various design modifications in order to arrive the methodology for better thrust vector control. A baseline CD rectangular nozzle model is considered and the modifications are done at the divergent portions by forming expansion regions at various points along with directional control. For the present investigation, four models have been chosen for simulation. The computational analysis is carried out with commercial CFD tool - Ansys Fluent which solves Density Based Navier Stokes (DBNS) equations. The solver uses implicit formulation and solves flow and turbulence equations by considering the fluid as ideal gas. A hypothetical domain outside the nozzle exit is considered to predict the nozzle jet plume. A different Nozzle Pressure is set up for all four configurations and the analysis is carried out for better convergence. CFD is able to predict the exit shock formations and Mach variations along the jet length effectively and the jet orientations are significantly affected by the design modifications done at the divergent portion.*

Key words: CFD, Thrust vector control, Deflection angle, CD Nozzle, NPR, etc.

1. INTRODUCTION

A de Laval nozzle (or convergent-divergent nozzle) is a tube that is pinched in the middle, making a carefully balanced, asymmetric hourglass shape. It is used to accelerate a hot, pressurized gas passing through it to a higher supersonic speed in the axial(thrust) direction, by converting the heat energy of the flow into kinetic energy. Because of this, the nozzle is widely used in some types of steam turbines and rocket engine nozzles. It also sees use in supersonic jet engines.

In supersonic nozzle, designed a conventional two dimensional nozzle is usually considered to consist of several regions they are the contraction in which the flow is entirely subsonic, the throat region in which the flow accelerates from high subsonic to a low supersonic speed, an initial expansion region where the slope of the contour increases up to its maximum value, the test section where the flow is uniform and parallel to the axis.

One of the more recent investigations in an experiment study of flow in rectangular over expanded supersonic nozzle exploring the complexity nature of such flows. Prediction of such flow also presence a great challenge to any CFD code.

Thrust vectoring also thrust vector control or TVC, is the ability of an aircraft, rocket, or other vehicle to manipulate the direction of the thrust from its engines or motors in order to control the attitude or angular velocity of the vehicle.

ANSYS-FLUENT can be used with Density Based Navier Stokes(DBNS) equation and the fixed NPR can be used with different configurations and the determine the mach variations along the exit of the nozzle for various deflection angle at the divergent portion of the nozzle.

2. CFD NUMERICAL STUDY

For the present comparison of all four nozzle configurations Computational Fluid Dynamics (CFD) is used. The numerical methodology followed during this analysis is explained below.

2.1 Physical Model

Four different configurations of CD Nozzle profiles have been considered for present numerical investigation. The first configuration is a baseline model with straight convergent and divergent passage. The second configuration is chosen in such a way that the flow is diverted from bottom side of the nozzle and the expansion is carried out in upward direction. The throat diameter is maintained as 22.9 mm and the consecutive flow diverting angles are 29.64º and 2.80º. The third configuration is chosen in such a way that the flow is diverted from both the top and bottom side of the nozzle divergent section and the flow diverting angles are 15º at top side and 8º at bottom side. In fourth configuration, the flow is diverted from both the top and bottom side of the nozzle and the consecutive flow diverting angles are 29.64º and 2.80º at both top and bottom side of the nozzle. Figure 1 shows the physical model of the first configuration used for the computational analysis. The outlet diameters of the nozzle for all four configurations vary with respect to the angle of flow diversions.
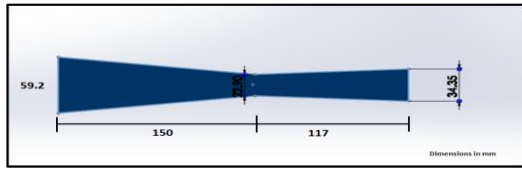

Figure 1: Nozzle Model for First Configuration

Figure 2 shows the physical model for second configuration used for the computational analysis.
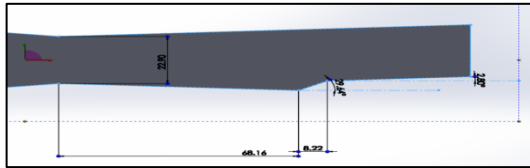

Figure 2: Nozzle Model for Second Configuration

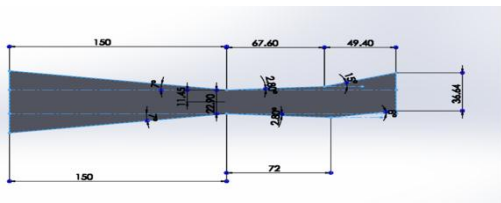Figure 3 shows the physical model for third configuration used for computational analysis.

Figure 3: Nozzle Model for Third Configuration

Figure 4 shows that the physical model for the fourth configuration for computational analysis
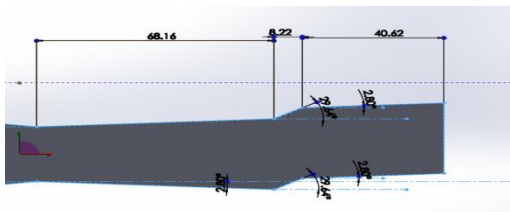

Figure 4: Nozzle Model for Fourth Configuration

2.2 Computational Model

The nozzle configurations are discritized with quad elements using a standard meshing tool (Hyper mesh). The computational domain is extended outside the nozzle in order to capture the jet core and plume. The domain size is chosen in such a way that the overall length of the flow domain is 12 times of the nozzle length.

Figure 5: Flow Domain for Present Numerical Simulation

The nozzle wall is given with a boundary layer mesh growth to capture the wall shear effects. The first node height from the nozzle wall is kept at 0.001 mm and the boundary layer mesh is generated by biasing the maximum number of nodes towards the wall sides. The boundary elements growth rate is around 1.2. Figure 6 shows the flow grid used for the present analysis.
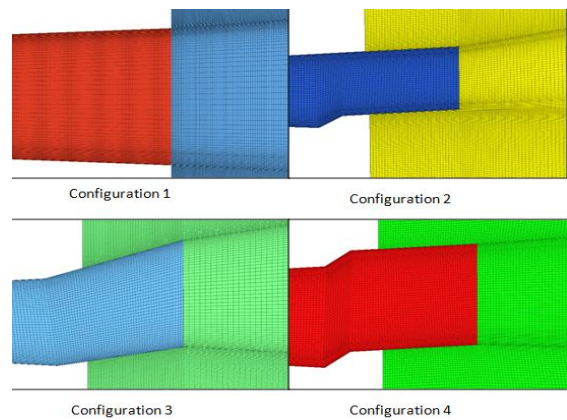

Figure 6: Mesh Refinements Near the nozzle wall regions

Figure 7 shows the Boundary mesh which is used to capture the wall shear and expansion near the throat section of the Nozzle
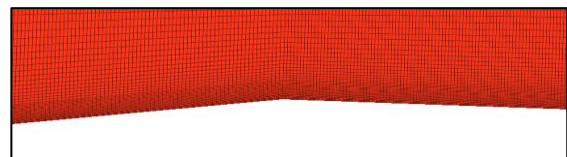

Figure 7:   Boundary mesh

2.3 Physics Definition

The discritized fluid domain is given with appropriate boundary conditions and solver settings to predict the nozzle jet plume and the simulation is carried out until the required flow patterns are arrived. For the present computational investigation a Density Based Navier Stokes (DBNS) solver (ANSYS – Fluent) is used and the flow is treated completely compressible. The fluid used for the analysis is Air- with ideal gas properties. The solver uses SIMPLE algorithm and finite volume method to solve the discritized equations. The flow equations (Continuity, Momentum & Energy) are solved along with Turbulence. For this simulation standard k-epsilon

turbulence model is used to predict the jet- plume formation outside the nozzle.

The nozzle inlet is given with 'Pressure Inlet' boundary conditions where the total pressure is mentioned as 5bar, 7bar. The nozzle inlet pressure can be varied from 3bar to 5bar for the both two configurations. The outlet condition is given with 'Pressure Outlet' boundary conditions where the guage pressure is given as 0 Pa. The reference conditions are taken with standard atmospheric conditions. The CFL number for the analysis is adjusted to match with the grid and the value assigned is 0.5.

3. RESULTS AND DISCUSSIONS

The converged solutions are post processed to analyze the effect of nozzle parameters in the divergent portion. Various flow parameters like Mach number, Pressure and density are compared for all four configurations.

Figure 8 shows the mach disc formation and jet plume decay for the baseline configuration. It can be noticed that the numerical simulation is able to predict the jet core correctly and the mach reductions are captured accordingly.
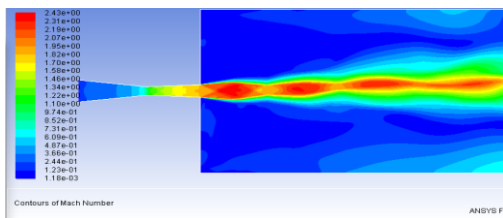

Figure 8: Jet Prediction of Baseline Configuration – Mach Contour

Figure 9 depicts the mach disc at the exit of the nozzle where the oblique shocks at the edge of the nozzle exit increases the Mach number and the shock pattern obtained are significantly matches with the actual plume pattern.
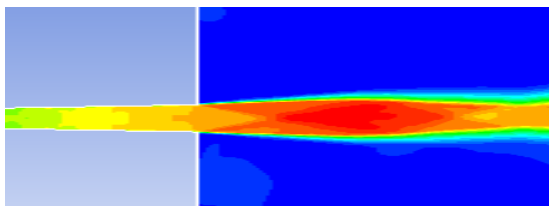

Figure 9: Mach Disc at the Exit of the Nozzle (Baseline)

Figure 10 shows the mach variations along the plume of nozzle configuration 2. It can be observed from the figure that the mach discs formed are diverted little upwards. In the baseline the maximum Mach number obtained is 1.70 where as in the nozzle configuration 2, it is predicted as 1.83.
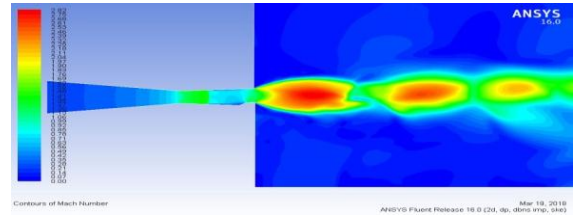

Figure 10: Mach Variations along the Plume –configurations2

Figure 11 shows the variation of density along the jet core. It can be observed from the contour that there is an intermittent pattern of density rise along the plume which directly indicates the shock train outside the nozzle.
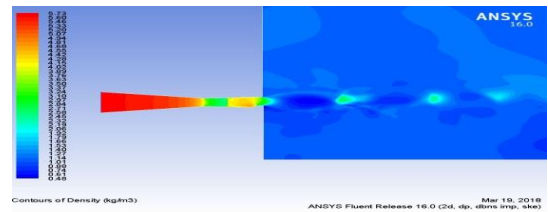

Figure 11: Intermittent density variations due to Shock Train formed inside the Jet Core

Figure 12 shows the mach variations along the plume of nozzle configuration 3. It can be observed from the figure that the mach discs formed are diverted upwards and produce significantly higher Mach number while comparing with the baseline configuration. In the baseline the maximum Mach number obtained is 1.70 where as in the nozzle configuration 3, it is predicted as 2.05.
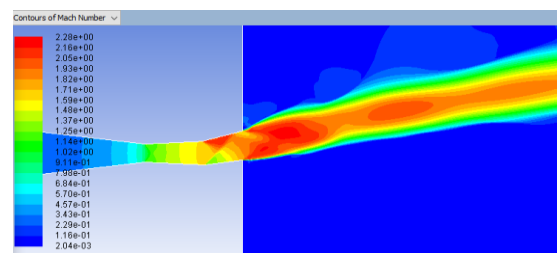

Figure 12: Mach Variations along the Plume –configuration 3

Figure 13 shows the mach variations along the plume of nozzle configuration 3. It can be observed from the figure that the mach discs formed are diverted upwards and produce significantly higher Mach number while comparing with the baseline configuration. In the baseline the maximum Mach number obtained is 1.70 where as in the nozzle configuration 4, it is predicted as 2.31.
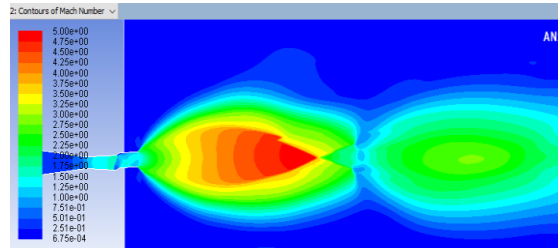
Figure 13: Mach Variations along the Plume – Configuration 4

Figure 14 shows the comparison of expansion process outside the nozzle exit. It can be interpreted from the figure that the flow diversions are significantly influenced by the geometric modifications done in the divergent portion. Also there exists a Mach number increment as the flow diverging angles are varied. This is because of the expansion fan generated at the diverging portion of the nozzle.
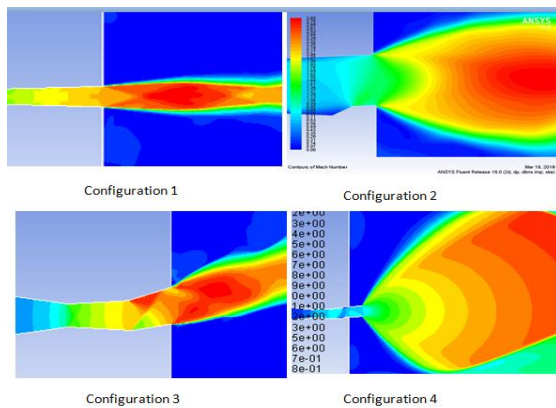

Figure 14: Comparison of Shock Pattern at the exit of Nozzle

Figure 15 shows the overall comparison of the exit Mach number obtained from the four nozzle configurations. It can be noted that the fourth configurations with both top and bottom side upward deflection angle produced higher Mach number while comparing with all other configurations. Also the base line nozzle produced lesser Mach number as the expansion fans are significantly influenced by flow turning angle.
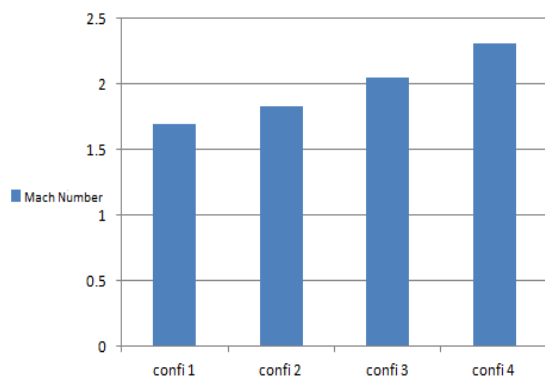

Figure 10: Exit Mach number Comparison

4 .CONCLUSION

The numerical fluid dynamics can be effectively used for the jet core prediction of a CD nozzle. The present investigations show that the plume diameter is higher in the second nozzle configuration. Also it can be observed from the results that the Mach number at the exit of the nozzle is lower in baseline configuration. Also it can be concluded that the configuration 4 predicts a higher Mach number when compared with other configurations.

REFERENCES

[1] Adamson T.C, Jr and Nicholls.J.A (1959) "On the Structure of Jets from Highly UnderExpanded Nozzles into Still Air," Published in Journal of the Aerospace Sciences, Volume.26, Jan 1959, pp. 16-24.

[2] Alrutz T & Knopp T (2007). "Near Wall Grid Adaptation for Turbulent Flows" published in International Journal Computing Science and Mathematics, Volume.1, pp:177-192.

[3] 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.

[4] Back L.H, Massier P & Cuffel R (1967). "Flow Phenomena and Convective Heat Transfer in a Conical Supersonic Nozzle" published in Journal of Spacecraft and Rockets, Volume.4, pp:8.

[5] Dutton J.C, "Swirling Supersonic Nozzle Flow," published in Journal of Propulsion and Power, volume.3, July 1987, pp. 342-349.

[6] E.M.S.Ekanayake, J.A.Gear and Y.Ding, (2010). "Numerical Simulation of a Supersonic Convergent Divergent Nozzle with Divergent Angle Variations for under Expanded Condition", proceedings of 17th Australasian Fluid Conference, Auckland, Newzealand, December 2010.

[7] E .L. Morrisette and T. J. Goldberg (1978). "Turbulent Flow Separation for OverExpanded Nozzles" published in NASA Technical Paper:1207.

[8] Flack, Ronald D (2005). "Fundamentals of Jet Propulsion with Applications" published in Cambridge University Press, June 2005, Doi:10.2277/0521819830, ISBN 978-0521819831.

[9] 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.