Turkish Online Journal of Qualitative Inquiry (TOJQI)
Volume 12, Issue 6, June 2021:246-260

Research Article

Design and Computational Analysis of Two-Dimensional Thrust Vectoring Convergent-Divergent Nozzle

Dr A Revanth Reddy[1], Rubesh Paramasivam[2*], Lawrence A[3], Dhanish Aswin S[4], Mr Kannan B T[5]

Abstract

Thrust vectoring or thrust vector control (TVC) has a large scope in research and development in the current and future design of various military aircrafts due to the higher maneuverability. This research paper is about design and simulation of 2D thrust vectoring convergent divergent nozzle for various angles of deflections. Vectoring thrust by using nozzle can be achieved by either mechanical or fluidic methods. This project is based on type two mechanical thrust vectoring where the divergent section of c-d nozzle is deflected in order to vector the thrust by using mechanical components such as two ring actuators system. A convergent divergent nozzle is designed according to literature survey using Ansys Design Modeler. A control geometry where the deflection angle of 0° (case 1) is designed and followed by the divergent section is then deflected 5°,10°,15°,20° downwards (case 2,3,4,5 correspondingly). A total of 5 geometries are then meshed using Ansys Mesh up to maximum quality and refinement for accurate results. The various cases are then analysed by keeping its boundary conditions same in all cases using Ansys Fluent. The result obtained for all cases are compared on how the fluid acts for various angles of deflection. Conclusions are drawn based on the graphical elements of contours, graphs and charts of velocity, pressure, mach

[1] School of Aeronautical Sciences, Hindustan Institute of Technology & Science, Chennai, India, arevanthr@hindustanuniv.ac.in
[2] School of Aeronautical Sciences, Hindustan Institute of Technology & Science, Chennai, India, rubesh9904@gmail.com
[3] School of Aeronautical Sciences, Hindustan Institute of Technology & Science, Chennai, India, lawrencea712@gmail.com
[4] School of Aeronautical Sciences, Hindustan Institute of Technology & Science, Chennai, India, dhanishaswin@gmail.com
[5] Department of Aerospace Engineering, SRM Institute of Science & Technology, Chennai, India

number and temperature. An inference is drawn as to the efficiency of deflection angle with minimum disturbance in fluid flow which will have a higher performance of thrust vectoring.

Keywords: *thrust vectoring, mechanical type, convergent-divergent nozzle, deflection angle of nozzle*

Introduction

Thrust vectoring is a crucial mechanical element in aircraft to obtain the change in angular velocity of the aircraft by deflecting the exhaust gas of the aircraft. This decreases the burden of the aircraft where conventionally the maneuverblity is obtained by utilizing the primary control surfaces such as aileron, elevator and rudder. Under special conditions, those primary control surface components will reduce the maneuverblity efficiency of the aircraft due to various issue. Thrust vectoring also can be optimized in taking off and landing of aircraft. Thus, this helps the minimize the distance of taking off and landing. Shorter distance allows more aircraft to take off and land in special terrains. In terms of ecology, this reduces the deforestation in order to build larger airport runways. CD nozzle is chosen due to its properties to produce subsonic airspeed condition as military aircraft are meant to react within a short span of time under supersonic condition. This paper discusses the mechanical aspects of thrust vectoring where the convergent and throat is fixed while the divergent section is deflected to a certain extend. As mentioned earlier, the exhaust gas is deflected and the direction of thrust changes. The study is done on the various flow property and shock lines effect.

1.1 Literature Survey

The author **Erich A. Wilson et al., (2000) [1]** has provided a real-life study of performance of a C-D nozzle during steady-state pitch vectoring and fundamental step for advanced aircraft under thrust vectoring implementation. This paper reviews the steady-state performances and the optimization observations initially obtained and the results are discovered. **Daniel Icaza (2000) [2**] has presented that ITP TVN design is a Convergent-Divergent axisymmetric round nozzle with 2D and 3D thrust vectoring, mechanically actuated, and where the deflection of the gas flow is achieved by deflecting the divergent section alone. **Markus Rutten et al., (2018) [3]** has developed efficient thrust vectoring devices for high ratio nozzles under the aspect of low radar and infrared observability. In this study, both the mechanical and the fluidic thrust vector control shows that can be integrated. It was also shown that the yaw control efficiency is high to be valuable for future projects. **Hao Wang, et al., (2019) [4]** has explained the fluid dynamics and the infrared radiation characteristics of the 2D-CD vectoring exhaust system with deflection angles from 0 to 20 and the results were compared with those of the base axisymmetric exhaust system. They have concluded that the fluid dynamic performance of a properly designed 2D-CD TV system is similar to conventional type. **Craig A Hunter et al., (2004) [5]** has performed the experimental study of separated nozzle flow. Result states that the test nozzle was influenced by shock induced boundary layer separation at overexpanded situation. The major transition is found to occur when the NPR is increased. **B.P. Madhu, et al., (2020) [6]** has designed the nozzle by keeping the convergent part constant and deflecting the divergent part by 9°, 11° & 13°. K-e model resulted in most proximate results to that of experimental. Absence of shocks was observed with higher divergent angle both in case of conical and divergent C-D nozzles.

1.2 Research Gap

The literature survey has explained the principle of thrust vectoring under the supersonic conditions. The research gap that we found was that many authors have used liquid injection for thrust vectoring and we wanted to check the effectiveness of a mechanical system for vectoring the thrust. If we deflect the divergent section only, how would be the flow properties be affected. In addition to that, we wanted to check the likelihood that how would the deflection would affect the flight performance when we need to achieve higher manuverbility while minimizing the flow disturbance. This method be could be used for STOL if conditions apply. This research gap will be mainly the idea for the project thus concluding the two-dimensional thrust vectoring CD nozzle under the need of supersonic conditions and performance during take-off and landing.

Geometries

The modelling is initiated with the geometry of the CD nozzle. The geometry is done by using the Design Modeler software embedded in Ansys Fluent. As the paper aims on thrust vectoring of CD nozzle, a total of 10 different geometries of CD nozzle case 1,2,3,4 & 5 consisting of with and without an external domain. The external domain is used to study the exhaust flow pattern and the fluid properties.

Table 1: Dimension of TVCD nozzle and its domain

Parameters	Dimensions
Total length of the nozzle	200 mm
Length of convergent section	66 mm
Length of divergent section	134 mm
Inlet diameter	60 mm
Throat diameter	20 mm
Divergent diameter	67 mm
Convergent angle	17 °
Divergent angle	10 °
Divergent deflected angle for Case 1	0 °
Divergent deflected angle for Case 2	5 °
Divergent deflected angle for Case 3	10 °
Divergent deflected angle for Case 4	15 °
Divergent deflected angle for Case 5	20 °
Length of the domain	600 mm
Height of the domain	300 mm

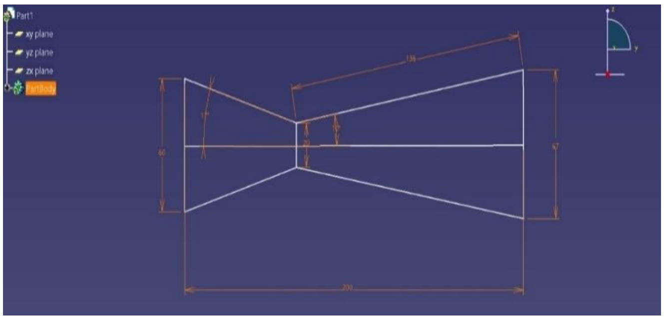

Figure 1: The dimensions marked at the CD nozzle as per in the Table 1 (Case 1)

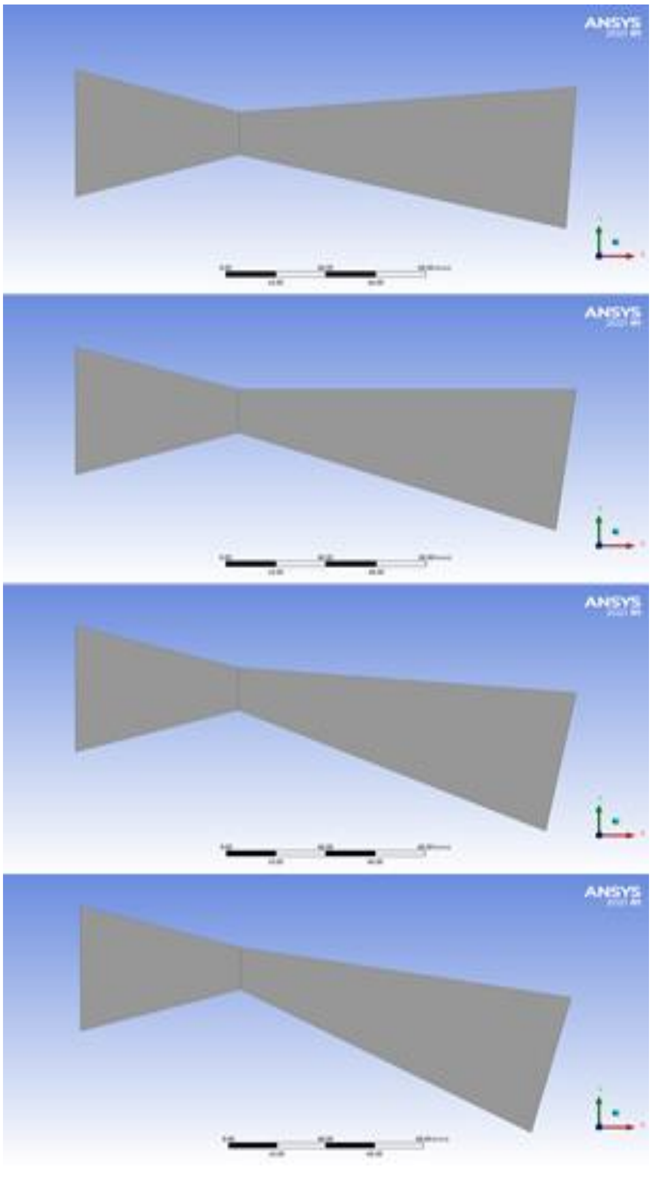

Figure 2: Geometries of Case 2,3,4 & 5 (top to bottom)

The convergent and throat section of the thrust vectoring CD nozzle is fixed and only the divergent section is deflected for various angles as stated in Table 1. The geometries of the nozzle that has been deflected downwards for the different cases is shown in Figure 2.

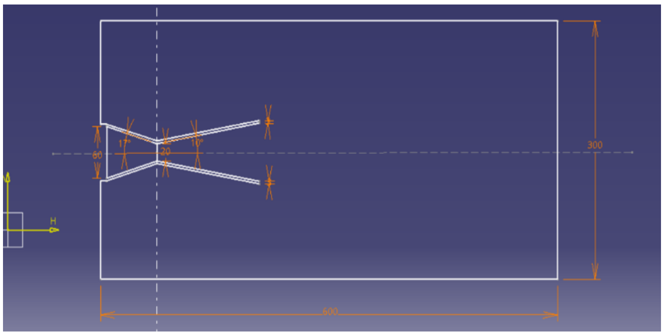

Figure 3: The dimensions marked at the CD nozzle with domain as per in the Table 1 (Case 1)

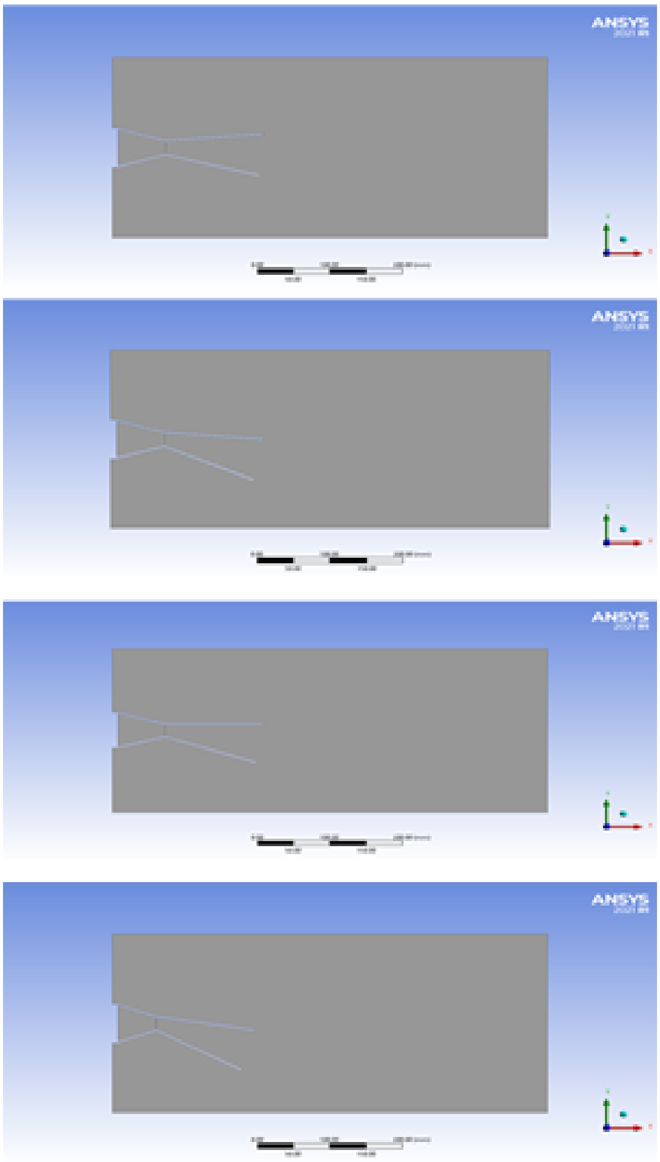

Figure 4: Geometries with domain of Case 2,3,4 & 5 (top to bottom)

The designed nozzle is further modelled with an external domain whose dimensions are shown in Figure 3. The deflection of the nozzle for the different angles as mentioned in Table 1 is shown in Figure 4.

Meshing

The next step in the process is the generation of mesh and the mesh of the standalone nozzle has been shown in Figure 5, followed by the mesh for the geometries with the external domain, which are shown in Figure 6. All the geometries are meshed using Ansys Mesh Generator.

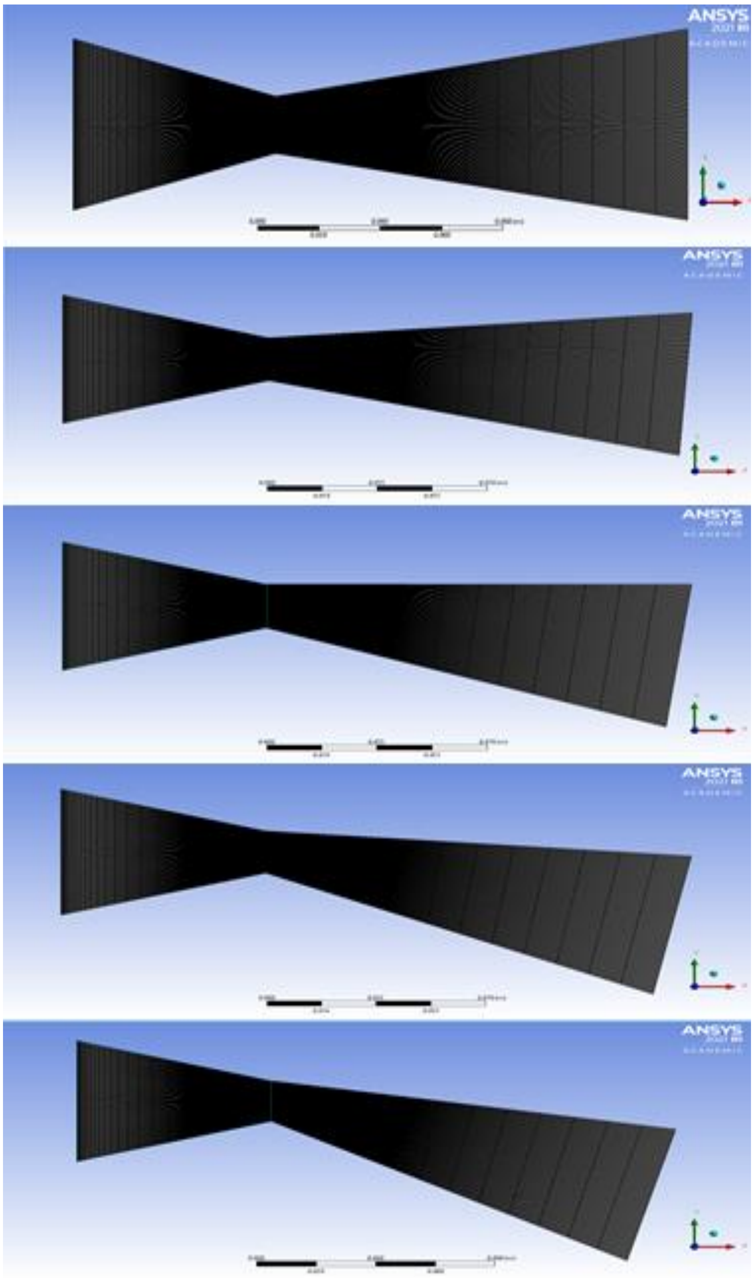

Figure 5: Meshed nozzle of the Case 1, 2, 3, 4 & 5 (top to bottom)

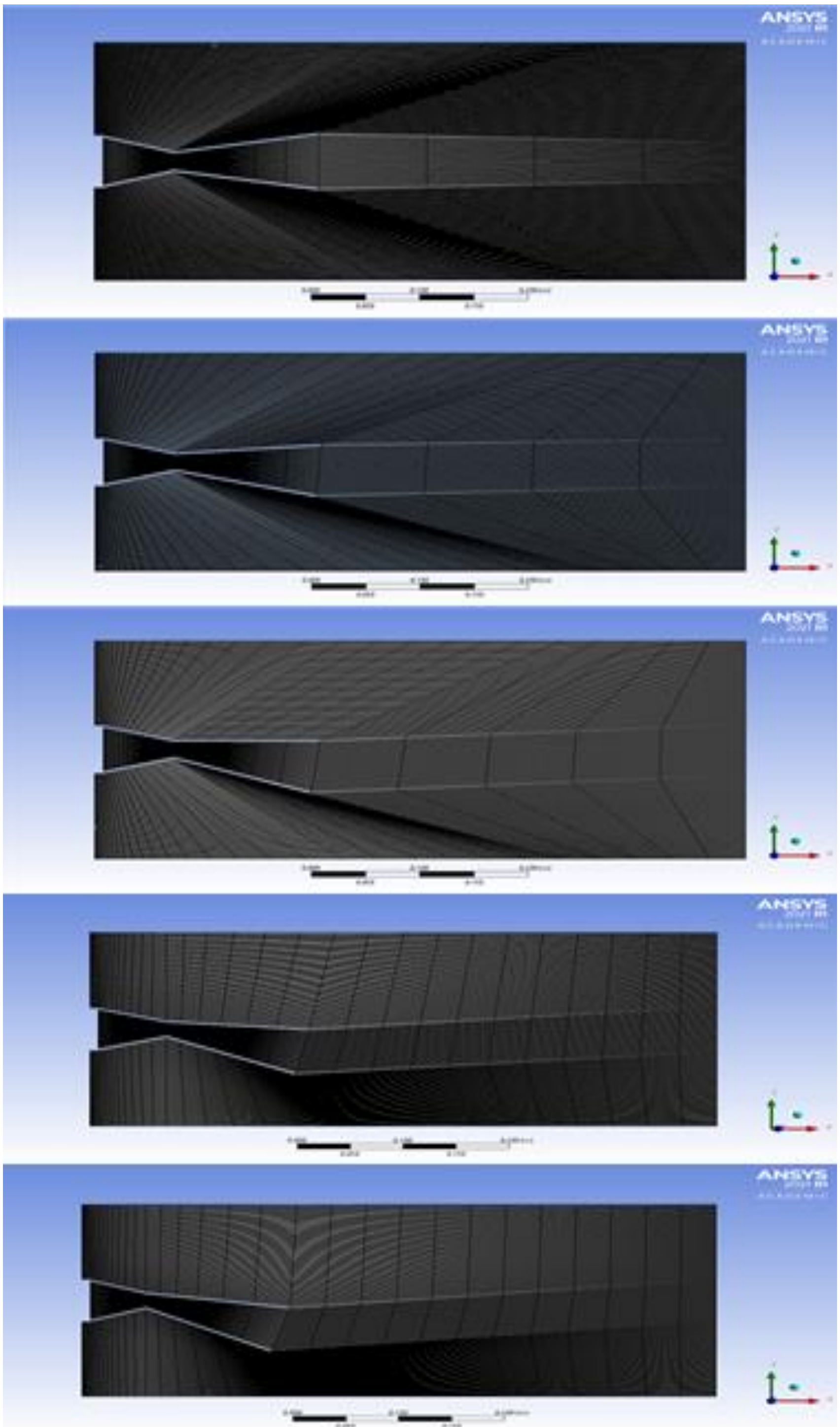

Figure 6: Meshed nozzle with domain of the Case 1, 2, 3, 4 & 5 (top to bottom)

Boundary Conditions

A density based, 2D space planar and K Epsilon viscous model is set as the solver in ANSYS FLUENT. The energy is given based on the heat transfer solution property. Under the materials option, the ideal gas is chosen together with the default properties of the ANSYS FLUENT. The inlet is chosen as pressure inlet and is given a value of 8 bar and temperature of 300K. The outlet is chosen as pressure far field and mach of 0.3. A standard initialisation is selected and the number iteration is set for 10000 or till the solution converges.

Result & Discussion

The Mach contours and the pressure contour of the nozzle with zero deflection is shown in Figure 7. This case is taken as the reference model for comparison with the nozzles having the different deflection angles which are shown from figures 8 through 11. It can be seen from the comparison that the mach and the pressure contours that there is a significant change in the flow pattern of the gases inside the nozzles as they are deflected and that there is a corresponding change in the pressure which indicates that a proper thrust vectoring effect has been achieved by deflecting the nozzles.

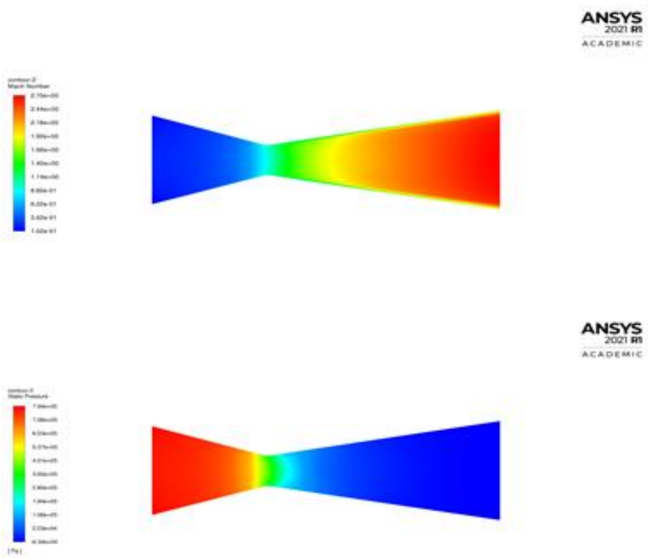

Figure 7: The mach (top) and pressure (bottom)are contours of Case 1 (0°) CD nozzle.

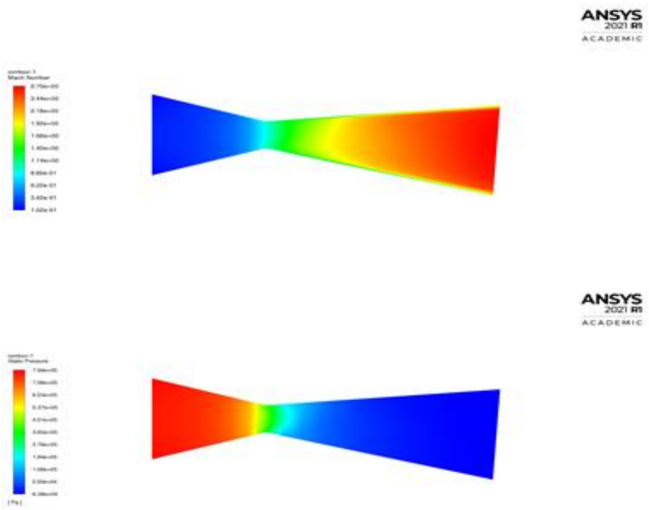

Figure 8: The mach (top) and pressure (bottom) are contours of Case 2 (5°) CD nozzle

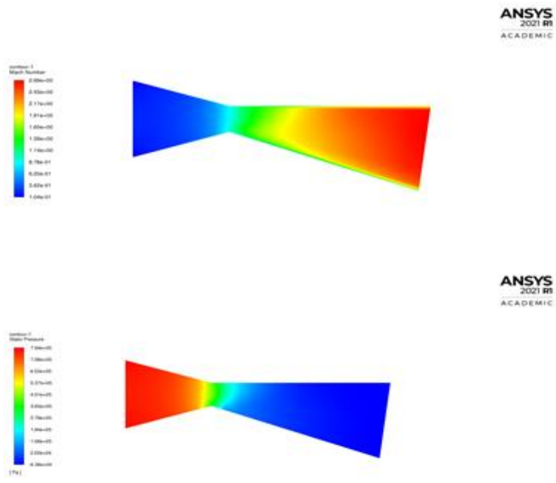

Figure 9: The mach (top) and pressure (bottom) are contours of Case 3 (10°) CD nozzle.

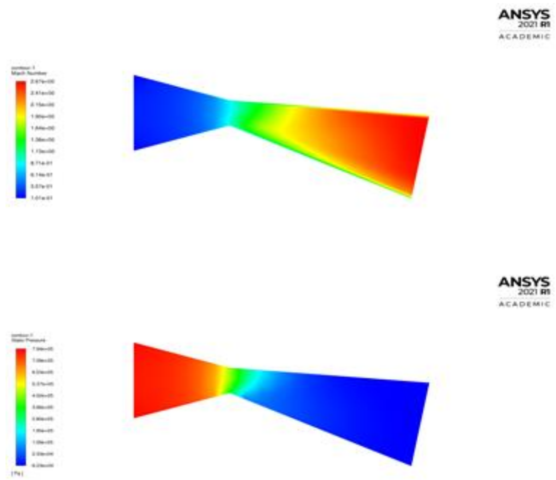

Figure 10: The mach (top) and pressure (bottom) are contours of Case 4 (15°) CD nozzle.

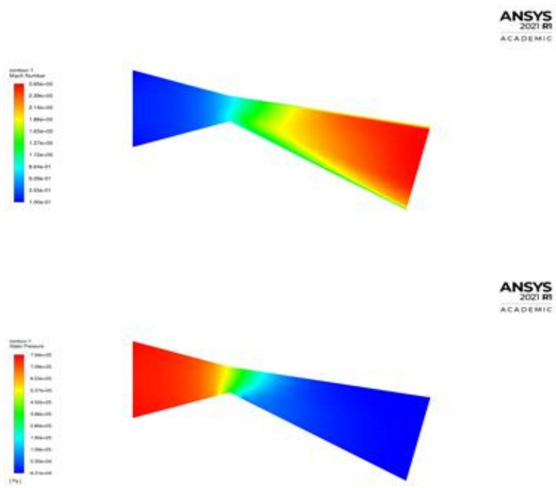

Figure 11: The mach (top) and pressure (bottom) are contours of Case 5 (20°) CD nozzle.

From the simulation which was done to study the formation of exhaust flow pattern and its characteristics, the results are shown in Figure 12. It can be seen that shocks are formed in the exhaust flow due to high back pressure. Oblique shocks that are formed at the outlet of the nozzle are clearly observed from the figures. The contour also shows the change in the shock structure that is formed as a result of the deflection of the nozzles at various angles. It can be seen that the shock is produced and deflected at the wall of the divergent section. The mach disc, expansion fan, oblique shock waves are also formed. It can be seen from the different contours of the flow that the change in the angle of deflection of the diverging section causes a disruption in the flow and the pressure and velocity experience a change accordingly. The pressure changes up to a maximum of 0.287%. The Mach number shows a reduction from case 1 to 5 by 2.12%.

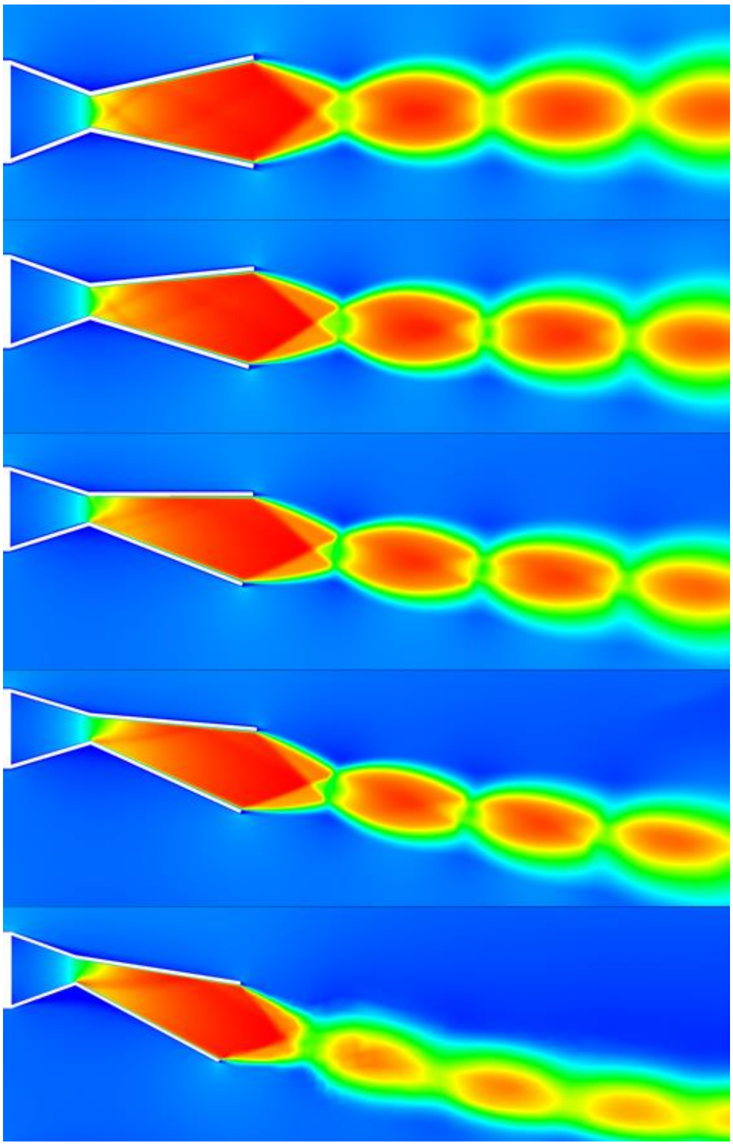

Figure 12: These are the contours of the exhaust flow pattern for the Case 1, 2, 3, 4 & 5 respectively

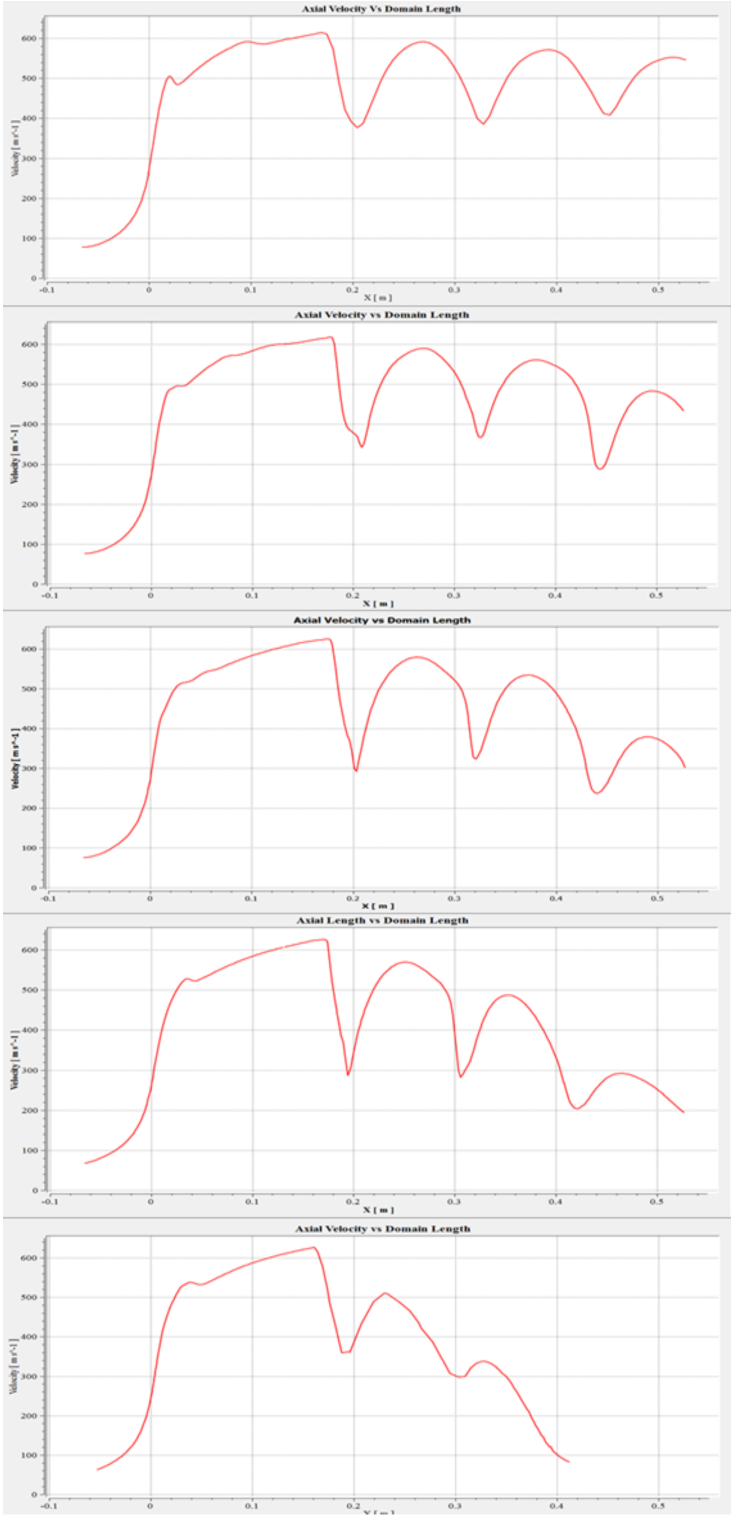

Figure 13: Graph above shows the axial velocity in Y axis and the domain length in X axis of Case 1, 2, 3, 4, & 5 respectively.

The change in the axial velocity of the gases after they exit the nozzle is shown in Figure 13. It can be seen that the velocity decreases as the gases expand fully in the axial direction. There is notable decrease and increase in the velocity of the gases as the shocks cross each other.

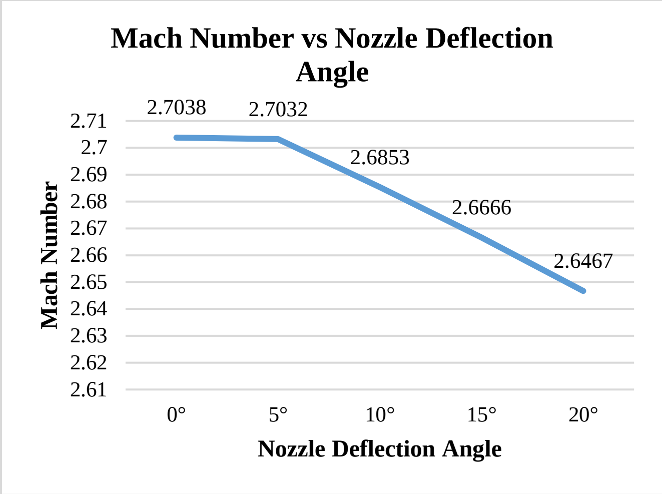

Figure 14: Graph of Mach Number vs Domain Length

The Mach no vs deflection angle has been plotted in Figure 14. We see a slight reduction in the mach no as the angle of deflection increases but it should be noted that the reduction is very small and doesn't affect the overall thrust that is generated by the nozzle by a huge margin.

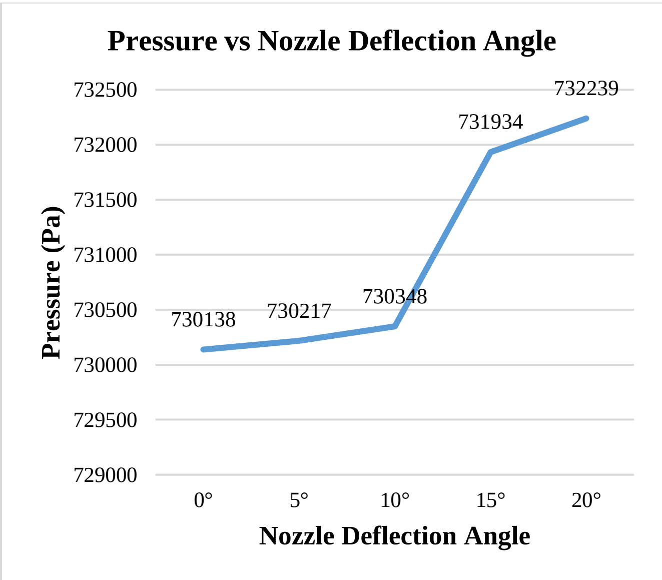

Figure 15: Graph of Pressure vs Domain Length

The Pressure vs deflection angle has been plotted in Figure 15. It can be seen from the graph that as the deflection angle is increased, the pressure at the exit of the nozzle increases by a small amount. This incremental gain in the exit pressure doesn't affect the overall thrust by a huge margin.

Grid Independent Study

Grid independent study is done validate the lowest involvement of the grid's influence on the calculation done by the ANSYS FLUENT Solvent. Under the same case (0° angle of deflection), the CD nozzle is subjected to increasing number of divisions and the resulting mach contours have been compared.

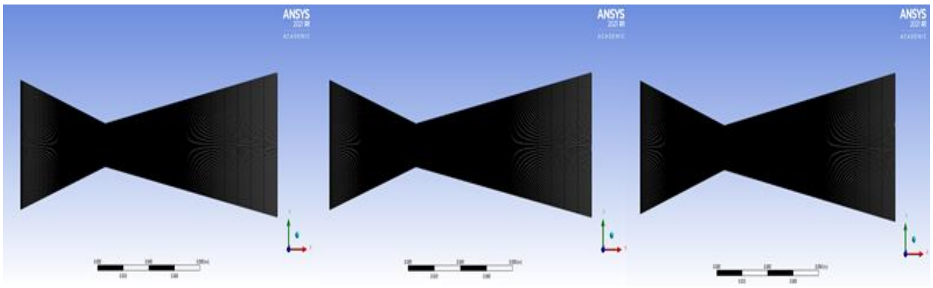

FIGURE 16: The number of divisions at 350 (right), 400 (middle), 450 (right)

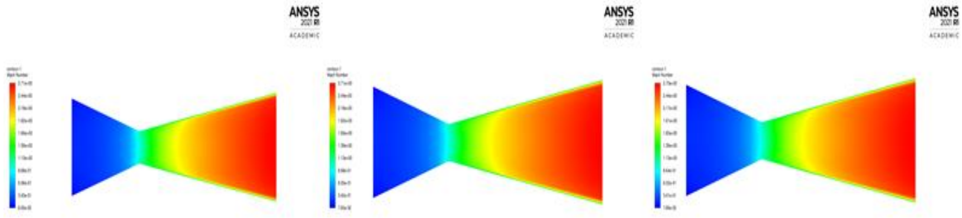

Figure 17: The mach contours of the meshing with number of divisions of 350 (left), 400 (middle), 450 (right) respectively

By comparing the mach contours in Figure 7 and Figure 17, we can see that Mach no magnitude is almost similar for all three cases. Thus, we can say that the quality of the grid used in this study give a satisfactory result irrespective of the number of divisions and it does not interfere with the simulation result

Conclusion

CFD analysis has been done on two-dimensional thrust vectoring convergent divergent nozzle. It has been found that the flow velocity is decreasing with increase in nozzle deflection angle. In domain simulation of 5°,10°,15°&20° the shocks in the divergent section get deflected according to the various cases. Initial increase shows the increasing velocity and drops down whenever a shock starts to get induced at the specific point. The deflection of shock wave in the divergent section causes imbalance of shock production as the line begins to direct upwards and less compared to downward as the angle of deflection increases. This has caused the disturbance in flow properties and creates resistance in pressure. Increased pressure has resulted in reduction in flow velocity. The maximum mach number (2.7038) is obtained at 0° deflection and minimum mach number (2.6467) at 20° deflection. Even though the disturbance has caused lower efficiency in flow properties, there is no big change in magnitude of thrust produced. Hence it has been found from the computational analysis that vectoring a CD nozzle by using mechanical components is achievable without any big loss in thrust produced.

Acknowledgements

We would like to thank Dr Parthasarathy Vasanthakumar for his suggestion on the research work. We would also like to thank our HOD and other professors of School of Aeronautical Sciences of HITS, Chennai who has been giving continuous contribution directly or indirectly throughout the research work. We are also deeply indebted to our parents and friends who have been the greatest support while we worked day and night for the project to make it a success..

References

George P. Sutton & Oscar Biblarz. (2001) Rocket Propulsion Elements *Wiley and Sons* 608.

Erich A Wilson & Dan Adler. (2000) Thrust Vectoring Nozzle Performance Modelling *Journal of Propulsion of Propulsion and Power***19** 1.

Markus Rutten. (2018) Numerical Investigation of Thrust Vectoring for a High-Aspect Ratio Nozzle of an Unmanned Combat Air Vehicle *Applied Aerodynamics Conference.*

Danial Ikaza. (2000) Thrust Vectoring Nozzle for Military Aircraft Engines *Industria de Turbo Propulsion S A.*

A. V. Ramana Rao & I. Akhil. (2016) CFD Analysis of CD Nozzle at Two Divergent Angles for Prediction of Mach Number (M<1, M=1 & M>1) And Flow Parameters of Newtonian Fluid by Using ANSYS FLUENT 14.0 *International Journal of Engineering Sciences and Research Technology*.

Hao Wang & Honghu Ji Haohao Lu. (2019) The Influence of Nozzle Deflection on Fluid Dynamic and Infrared Characteristics of a Two-Dimensional Convergent–Divergent Vectoring Exhaust System *Journal of Aerospace Engineering.*

Craig. A. Hunter. (2004) Experimental Investigation of Separated Nozzle Flow *Journal of Propulsion and Power* **20** 3.

B P Madhu, G Mahendramani & K Bhaskar. (2020) Numerical Analysis on Flow Properties in Convergent-Divergent Nozzle for Different Divergence Angle *Online Journal.*

Jun Lin Chen & Ying Hao Liao. (2020) Parametric Study on Thrust Vectoring with A Secondary Injection in Convergent-D Divergent Nozzle *Online Journal.*

Mohammed Nishat Akhtar & Elmi Abu Bakar. (2019) Numerical Simulations of a CD Nozzle and the Influence of Duct Length*International Journal of Innovative Technology and Exploring Engineering* **8** 9S2.

Abhilash Patil Aswatha Narayan & Rajashekara. (2019) CFD Analysis of Thrust Vectoring Of C-D Nozzle *International Journal of Latest Engineering Science* **2** 3.

K. Nehru & T. Kumarasan (2015) Computational Study of Mechanical Thrust Vectoring *International Journal of Research in Aeronautical and Mechanical Engineering* **3** 10.

K.P. Surya Narayana & K. Sadhashiva Reddy (2016) Simulation of Convergent Divergent Rocket Nozzle Using CFD Analysis *IOSR Journal of Mechanical and Civil Engineering* **13** 4.

Gutti Rajeshwara Rao & U.S. Ramakanth (2013) Flow Analysis in A Convergent-Divergent Nozzle Using CFD *International Journal of Research in Mechanical Engineering* **1** 2.

G Satyanarayana&C Varun (2013) CFD Analysis of Convergent-Divergent Nozzle *Acta Technica Corviniensis*.

Sasa Živkovic & Momcilo Milinovic. (2014) Experimental and Numerical Rest of a Supersonic Planar Thrust Vectoring Nozzle Via Mechanical Tabs.