International Journal of Latest Engineering Science (IJLES) E-ISSN: 2581-6659
Volume: 02 Issue: 03 May to June 2019 www.ijlesjournal.org

CFD Analysis of Thrust vectoring of C-D Nozzle

Abhilash Patil[#1], Aswatha Narayan[#2], Rajashekara V[#3]
[#1]*M. Tech AerospaceEngineering (Student)IIAEM-Jain University, Bangalore (India)*
[#2] *Aerospace Engineering (Professor)IIAEM-Jain University, Bangalore (India)*
[#3]*M. Tech Thermal Engineering (Student) Global Academy of Technology, Bangalore (India)*

ABSTRACT
Thrust Vectoring could be a comparatively new technology, and a number of other programs worldwide have explored its application and advantages. Thrust Vectoring will offer trendy military aircraft with variety of benefits relating to higher maneuvering, performance and survivability, all of that has associate degree influence upon Life Cycle price. There are many kinds of Thrust Vectoring Nozzles. There are other ways to attain the deflection of the gas jet: the most efficient one is by mechanically deflecting the divergent section only, hence minimizing the impact on the engine upstream of the throat section. This paper focus on designing a Convergent-Divergent nozzle with various cases of vectoring and non-vectoring to attain supersonic flow and optimizing it to achieve maximum drop pressure, Mach number. The analysis is performed according to the Convergent-Divergent nozzle with various cases i.e. mechanically deflecting the divergent section downward by an amount 5°, 10°, 15° and 20° angles over an axis keeping the same boundary condition. Our objective is to investigate Pressure drop, Mach number for the various cases of vectoring and non-vectoring Convergent-Divergent nozzle using CFD. The baseline Convergent-Divergent Nozzle i.e. non-vectoring nozzle analysis is carried out using various turbulence model (Enhanced k-epsilon, Spalart–Allmaras, SST k-omega) to investigate the impact of turbulence models on Pressure drop and Mach number. From this study it reveals that, the baseline case i.e. Convergent-Divergent nozzle with non-vectoring using Spalart-Allmaras turbulence model with 5% Turbulence Intensity resulted into less pressure drop (12.83 %) as compared to k-epsilon (12.98%) and k-omega (14.10%) models. Attained maximum supersonic flow with exit Mach number using Spalart-Allmaras model is 2.07.

Further, CFD analysis is conducted on vectoring cases of Convergent-Divergent nozzle i.e. downward deflection of divergent portion of nozzle (5°, 10°, 15° and 20° angles) using Spalart-Allmaras turbulence model. It has been found that Convergent-Divergent nozzle gives to increase Mach number. So, thrust vectoring applicable in Convergent-Divergent nozzle. Further deflecting with various angles thrust vectoring will be varying. In downward defection of thrust vectoring nozzle, 10° gives maximum increased in exit Mach number i.e.2.1251 which is 2.65% more than baseline case (non-vectoring nozzle). The appreciable amount of increased in the exit Machnumber is seen in vectoring cases compared to non-vectoring case of Convergent-Divergent nozzle.

Keywords: *CFD, Thrust Vectoring, C-D Nozzle, Turbulence Models*

I. INTRODUCTION

Vectoring is that procedure builds the exhaust nozzle structure intercommunicate make forward, Vertical or side-to-side thrust. Vectoring provides the directional thrust necessary for vertical takeoff and landing (VTOL) and short takeoff and landing (STOL) military aircraft. Vectoring additionally offers aircraft a more robust rate of climb and increase control during flight. n a military engine with heat up (afterburner), the nozzle features a confluent section, that is specially to accelerate the gas to supply thrust, nevertheless with the characteristic that it should be capable of varied the throat area (At) per the necessity of the engine running purpose. These are called “variable geometry convergent” nozzles. To boot, some nozzles embody a divergent section Downstream of the convergent section, that Over expands the jet between the throat area (At) and therefore the exit area (Ae) to additionality nevertheless some extra thrust. These are called “Variable geometry convergent divergent “nozzles.

From the purpose of read of the sort of deed suggests that, Thrust vectoring is classified:

• Fluidic Actuation: The deflection of the gas flow is achieved by injection of secondary airflows. This sort is particularly appropriate for fixed-area high expansion nozzles, like those utilized in rockets and missiles.

• Mechanical Actuation: The deflection of the gas flow is achieved by mechanical movement of the nozzle, that is supercharged by hydraulic or pneumatic actuators. This sort is particularly appropriate for variable geometry military aircraft nozzles.

One usually used thrust vectoring methodology involves manipulating the physical configuration of the exhaust nozzle on the jet engine or rocket motor. Once the nozzle is Physically turned to purpose in a very totally different direction, the thrust is additionally pointed during this new Direction This method is commonly used on aircraft and is the most typical methodology used with rocket-propelled launch vehicles.

Convergent-divergent nozzles are subtle items of hardware that are sometimes tuned to control below specific pressure and flow conditions. sterilization the nozzle's pure mathematics, specifically the pure mathematics of the mechanics throat, in an endeavor to vector the flow will have unwitting consequences that negatively impact engine performance and thrust.

In 1990, Berrier and Taylor [6] projected that employing a gimbal mechanism upstream of thenozzle to show the flow would have very little impact on the thrust. By turning the flow within the low-speed, subsonic region previous the throat, flow-turning losses are decreased. Their experimental information supports this claim. At mechanically deflected vector angles up to 25◦, and for avarious range of nozzle pressure ratios, the authors claim a maximum thrust loss of 2%. to boot, they surmise that this loss is perhaps thanks to a calculation error on their half. [7]

Turbulence model’ssummary:K-epsilon (k-ε)turbulence model is that the most famous used model in CFD. It's a 2-equation model that provides a general description of turbulence by suggests that of two transport equations (PDEs). k-epsilon model was to spice up the mixing-length model, still on perceive alternate to algebraically prescribing turbulent length scales in moderate to prime quality flows. The first transported variable is that turbulence kinetic energy (k). The second transported variable is that rate of dissipation of turbulence energy (ε).

k-ε model has been tailored specifically for planate shear layers and recirculation flows.

This model is that the foremost usually used and valid turbulence model with applications ranging from industrial to environmental flows, that explains its quality. it's sometimes helpful for free-shear layer flows with comparatively tiny pressure gradients yet as in confined flows wherever the Reynolds shear stresses are most significant. It may be explicit because the simplest turbulence model that solely initial and/or boundary conditions must be equipped.

However, it's costlier in terms of memory than the blending length model because it needs 2 additional PDEs. This model would be associate degree inappropriate alternative for issues like inlets and compressors as accuracy has been shown through an experiment to be reduced for flows containing giant adverse pressure gradients [citation needed]. The k-ε model additionally performs poorly in a very kind of necessary cases like unconfined flows, curvilineal boundary layers, rotating flows and flows in ellipsoidal ducts.

k–omega (k–ω) turbulence model is a2-equation turbulence model.The model makes an attempt to predict turbulenceby 2 partial differential equations for 2 variables, k and ω, with the primary variable being the turbulence kinetic energy (k) whereas the second (ω) is the specific rate ofdissipation (of the turbulence K.E. k into internal thermal energy).SST (Menter’s Shear Stress Transport) turbulence model could be a wide used and sturdy two-equation eddy-viscosity turbulence model utilized in Computational Fluid Dynamics. The model combines the k-omega turbulence model and K-epsilon turbulence model such the k-omega is employed within the inner region of the physical phenomenon and switches to the k-epsilon within the free shear flow.

The **Spalart–Allmaras** model is a one-equation model. The Spalart–Allmaras model shows better results in the wall-bounded flows and for boundary layers subjected to adverse pressure gradients.

II. 3-D MODELLING OF C-D NOZZLE

Our present CFD analysis is based on 3-D Mechanical thrust vectoring. The analysis is performed according to the C-D nozzle with various cases i.e. mechanically deflecting the divergent section downward direction by an amount of 5,10,15,20 degrees over an axis.

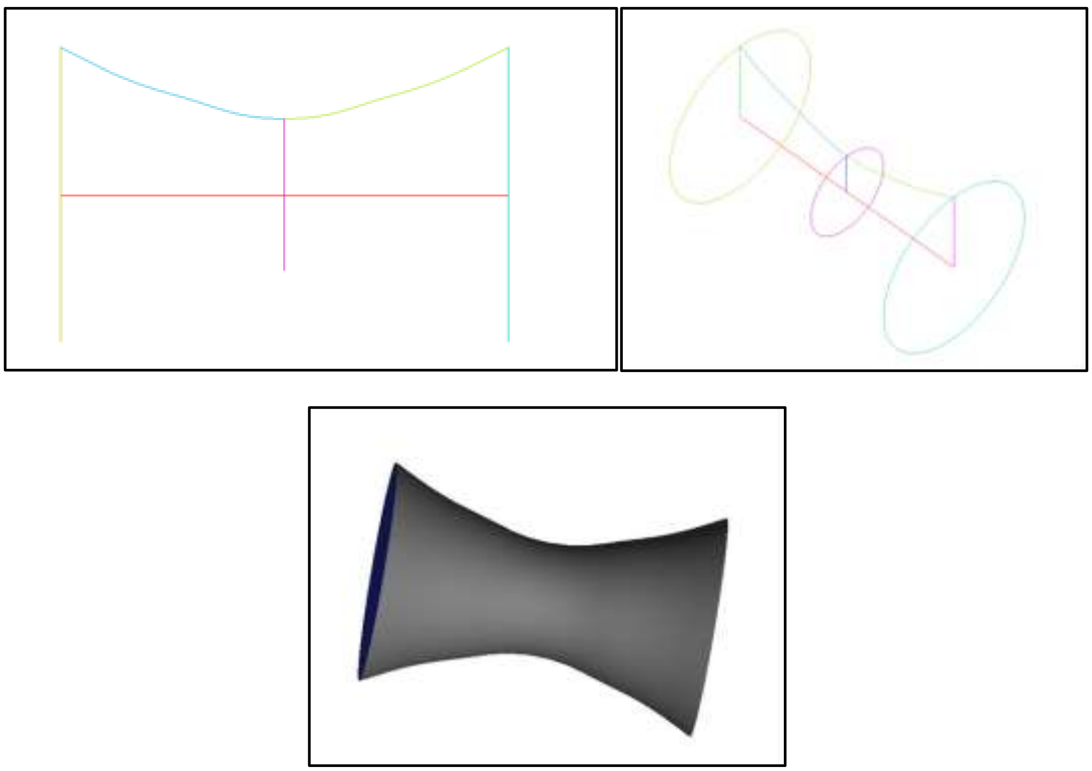

Figure 2.1: Geometry of Non-vectoring C-D nozzle

X	Y
-0.5	0.33
-0.4	0.28
-0.3	0.24
-0.2	0.21
-0.1	0.18
0	0.17
0.1	0.18

0.2	0.21
0.3	0.24
0.4	0.28
0.5	0.33

Table 2.1: Co-ordinates of nozzle profile[3]

The figure 2.2 shows the comparisons of baseline geometry vs. vectoring geometry (Deflected divergent portion of nozzle in downward direction by an amount of 5, 10, 15 and 20 degrees).

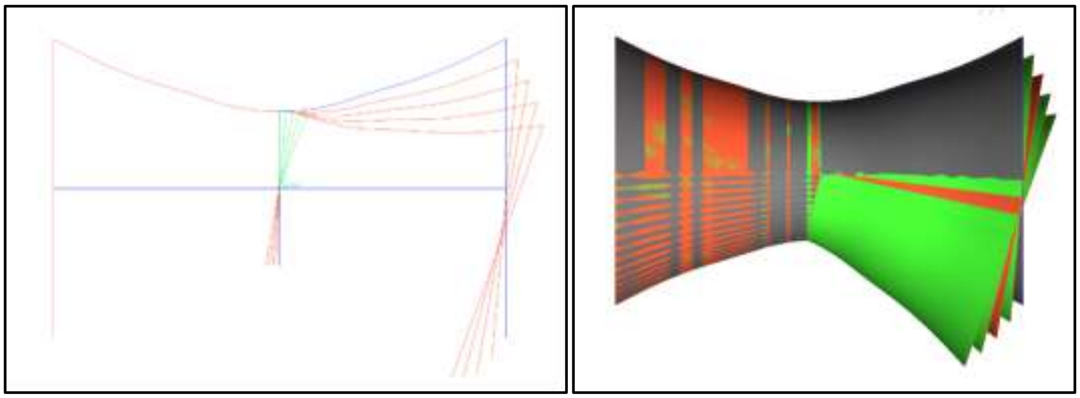

Figure 2.2: Comparison of Non-vectoring Vs. Vectoring C-D nozzle geometry

The mesh is generated using ICEM CFD software tool. The hexahedral structured mesh is generated with boundary layer. Adequately captures the near wall regions. O-grid block topology is used to generate mesh. The mesh size is ~0.5 million cells. The below figures show the mesh views at cross sectional plane, inlet and outlet. The same mesh quality is maintained for other vectoring cases of C-D nozzle.

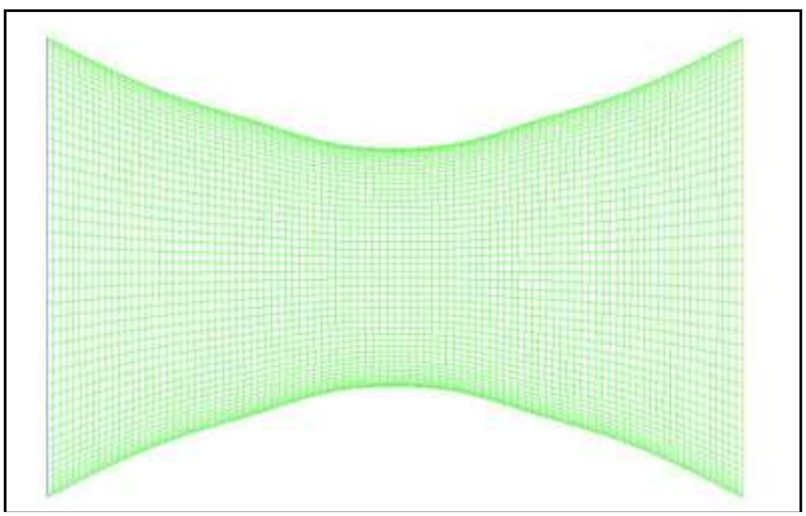

Figure 2.3: cross-sectional view of mesh.

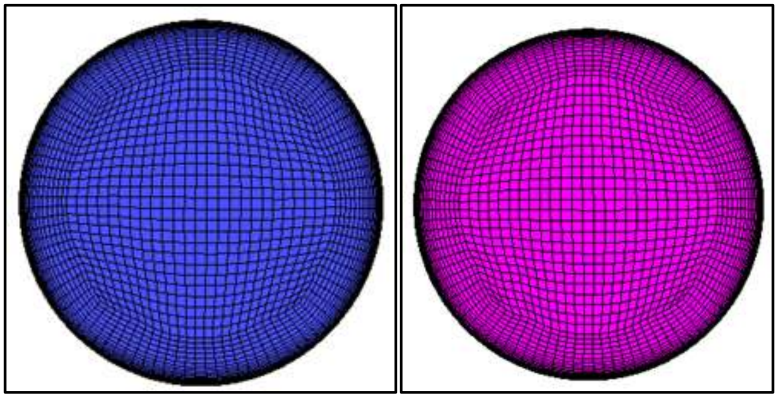

Figure 2.4: Inlet and Outlet mesh.

III. BOUNDARY CONDITION

Input used for CFD analysis is shown in below table 3.1.

Inlet Pressure (bar)	80
Inlet Temperature (K)	1583
Exit Pressure (bar)	13
Back flow Total Temperature (K)	1200
Equation of state	Ideal gas
Molar Mass (Kgk/mol)	23.05
Specific heat Capacity (J/KgK)	2034.6
Dynamic Viscosity (Kg/ms)	6.07e-05
Thermal Conductivity (W/mK)	0.0706
Turbulent Intensity (%)	5
Hydraulic Diameter (m)	0.66

Table 3.1: Input to CFD.

Pressure-Velocity coupling with coupled scheme is used.Spatial Discretization is Second order. At the starting, the solutions are initiated with lower under relaxation factors, and once solution get stabilized, then solution is switched over to default under relaxation factors i.e. higher relaxation factors.

IV. RESULTS AND DISCUSSION

4.1.Turbulence model sensitivity study for Baseline C-D nozzle (non-vectoring)

The analysis is carried out using SA, SST K-Omega, K-Epsilon enhanced wall treatment Turbulence models, to investigate the impact of turbulence quantities on Pressure drop and Exit Mach number for baseline C-D nozzle (Non-vectoring C-D nozzle). Various Turbulence model sensitivity studied i.e. k-epsilon and k-omega and SA is conducted with the 5% Turbulence Intensity and results are shown in below.

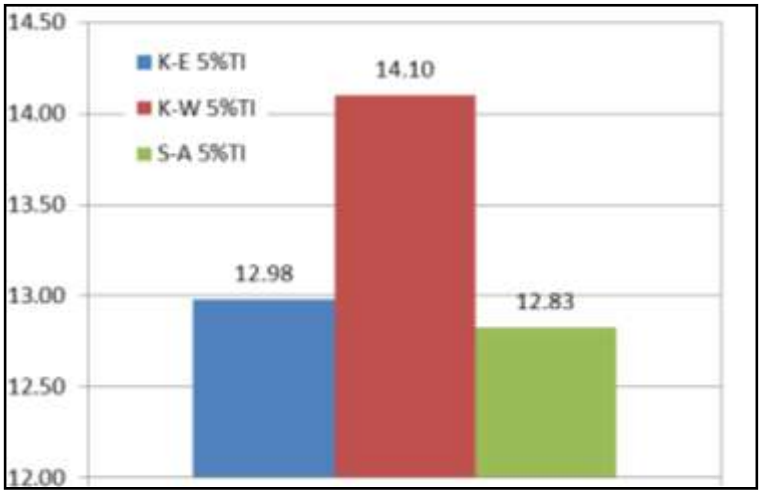

Figure 4.1.1: Pressure loss in percentage (Non-vectoring C-D nozzle).

The spectra shown in figure 4.1.1 and 4.1.2 indicates the pressure loss and exit Mach number for various turbulence models. The pressure loss for k-epsilon, k-omega and SA Turbulence models are 12.98%, 14.10% and 12.83% respectively. The exit Mach number for k-epsilon, k-omega and SA Turbulence are 2.0423, 2.04 and 2.07.

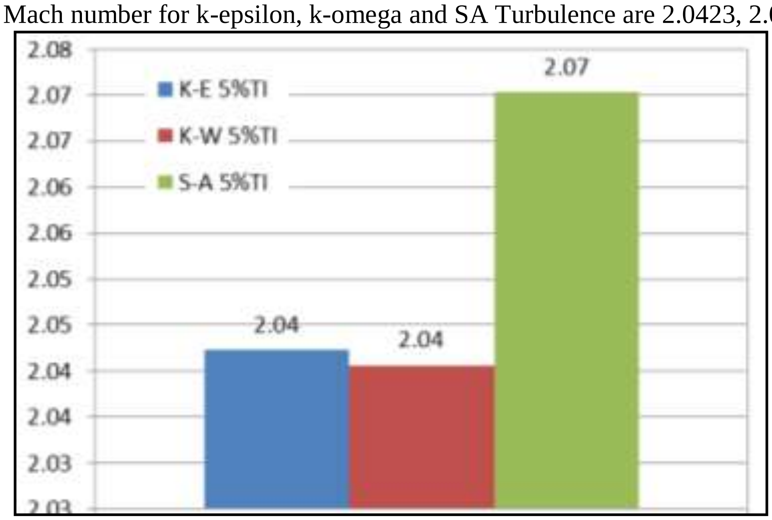

Figure 4.1.2: Exit Mach number (Non-vectoring C-D nozzle).

The baseline case i.e. C-D nozzle with non-vectoring using S-A turbulence model with 5% Turbulence Intensity resulted into less pressure drop i.e. 12.83 % as compared to k-epsilon and k-omega models. On-vectoring has attained maximum supersonic flow with 2.07 Mach number using S-A model with 5% Turbulence Intensity. From above figure 4.1.1 and 4.1.2, it is concluded that SA turbulence model yields less pressure drop and hence increased exit Mach number compared to k-epsilon and k-omega model.

The Further analysis is performed according to the C-D nozzle with various cases i.e. mechanically deflecting the divergent section downward by 5,10,15,20 degrees over an axis using SA Turbulence model with 5% Turbulent Intensity.

4.2. Mechanically Downward Deflected divergent section of C-D nozzle by 5,10,15,20 degrees

In this section, the results discussion of baseline case (non-vectoring C-D nozzle) and downward deflected divergent section of nozzle by 5, 10, 15 and 20 degree is presented. Also, we compared results of baseline case Vs downward deflected divergent nozzle cases (vectoring C-D nozzle). Pressure and Mach number contours plot are presented for all the cases of C-D nozzle.

Degree	Total Pressure Exit (Pa)	Mach Number	Pressure loss (%)
0	6973871.6	2.0702	12.8266
5	5816183.4	2.1249	27.2977
10	5793056.4	2.1251	27.5868
15	5800651.3	2.1233	27.4919
20	5695161.1	2.1052	28.8105

Table 4.2.1: Total Pressure, Mach number and Pressure loss

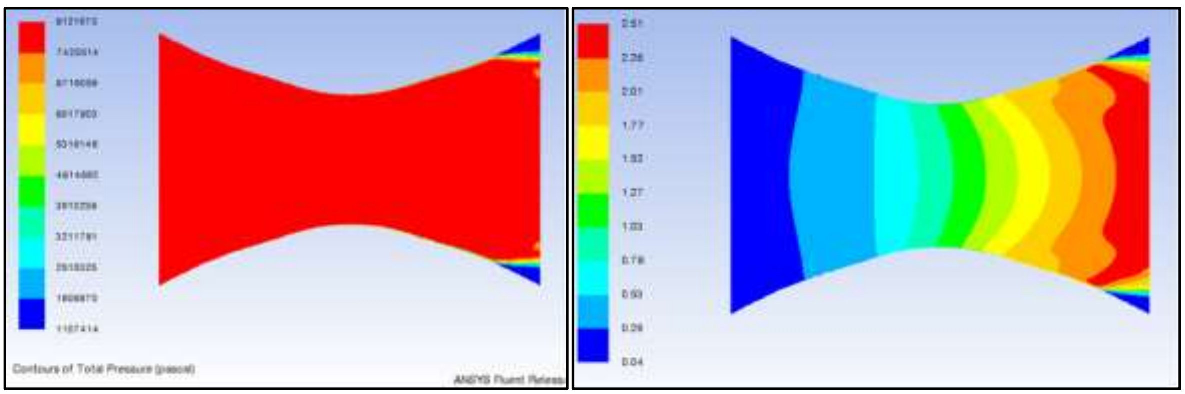

Figure 4.2.1: Contour of Total Pressure, Mach number for 0 degree

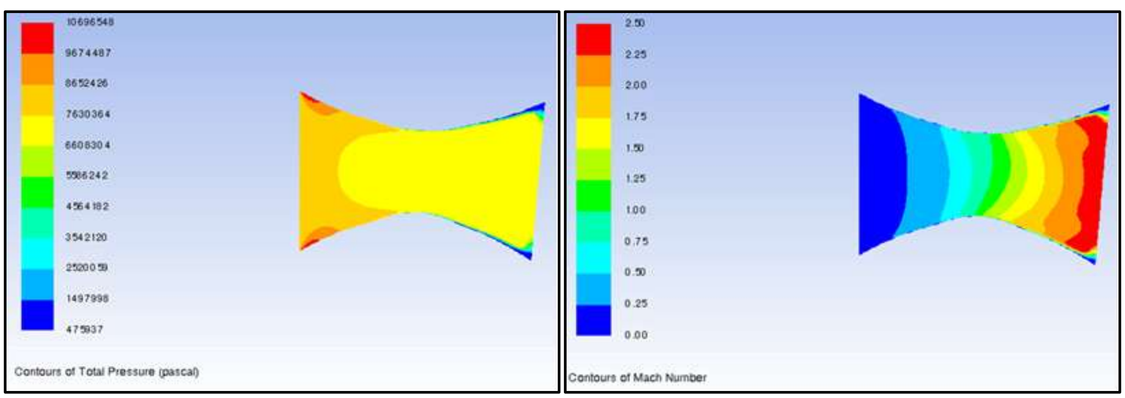

Figure 4.2.2: Contour of Total Pressure, Mach number for 5 degree

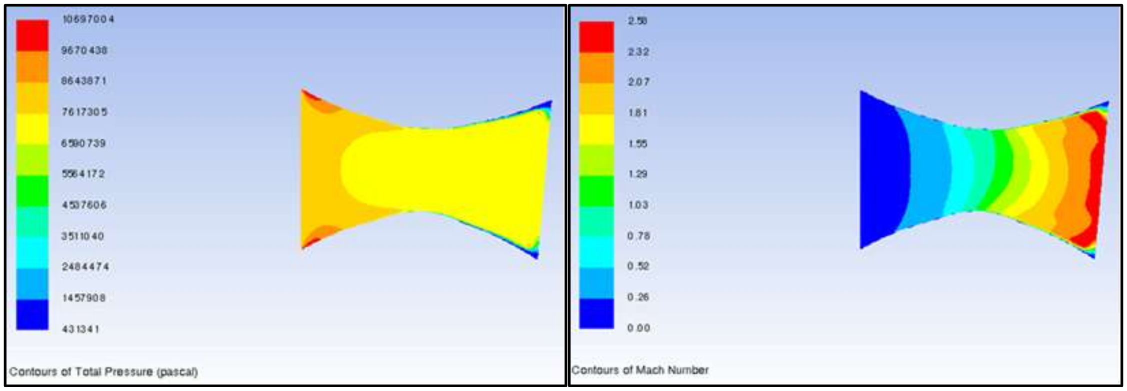

Figure 4.2.3: Contour of Total Pressure, Mach number for 10 degree

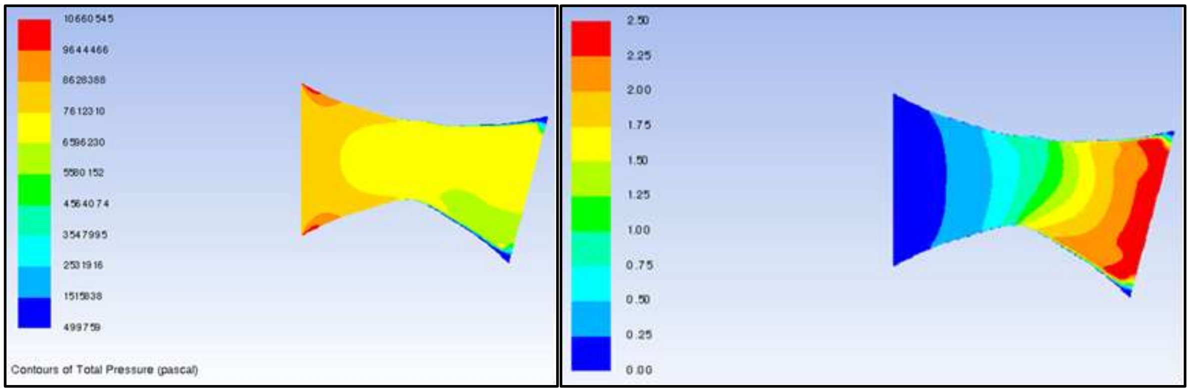

Figure 4.2.4: Contour of Total Pressure, Mach number for 15 degree

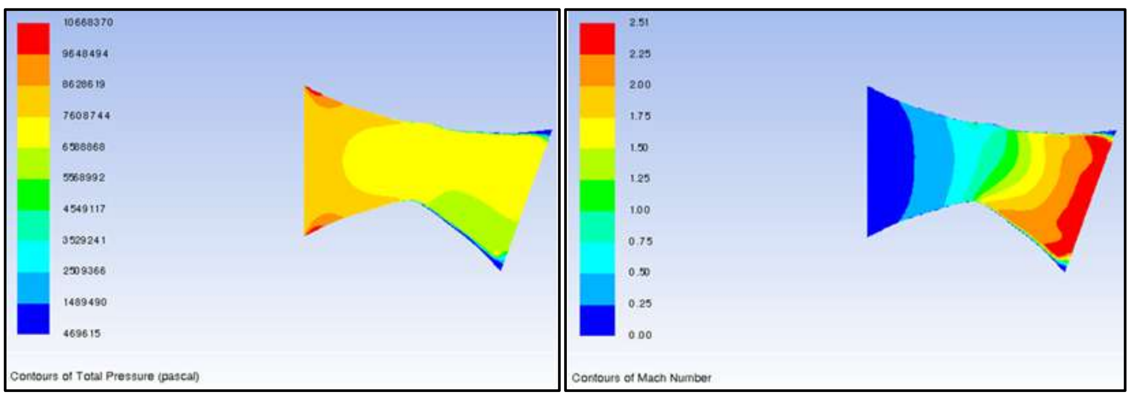

Figure 4.2.5: Contour of Total Pressure, Mach number for 20 degree

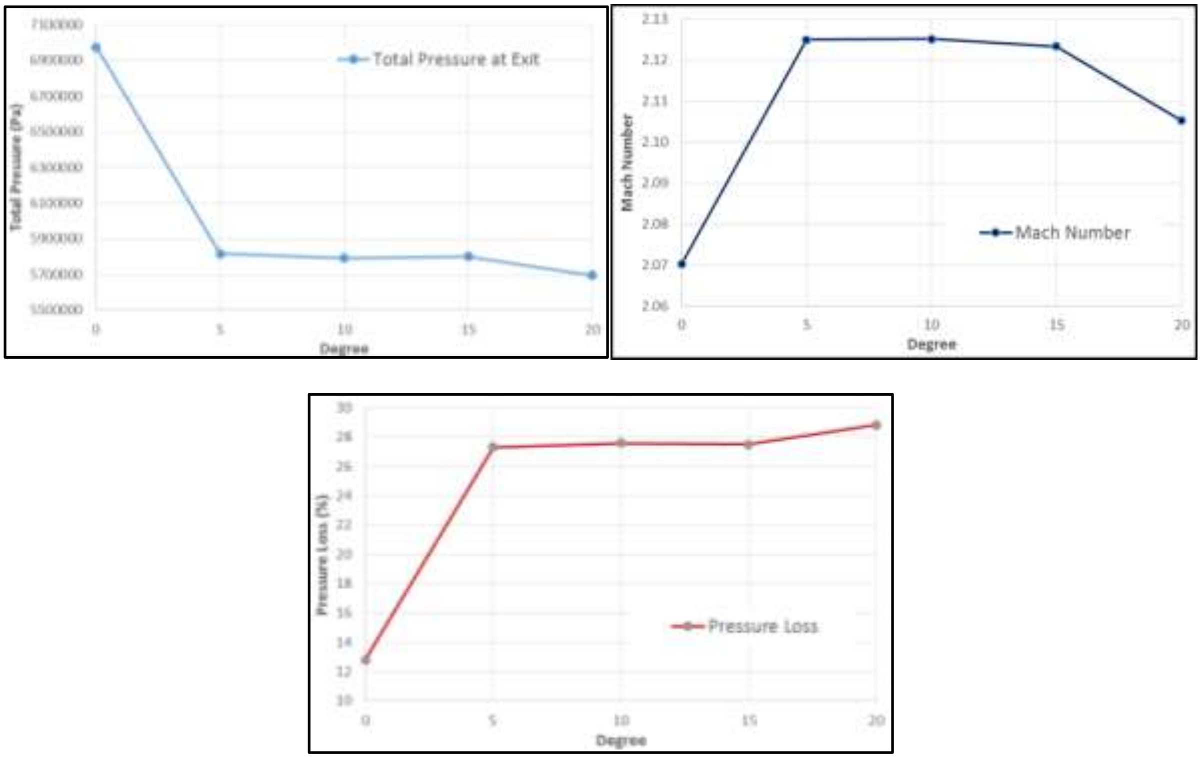

Figure 4.2.6: Comparison plots of Total Pressure, Mach number at exitand Pressure Loss

The figure 4.2.6 shows Mach number, Total Pressure exit and Pressure Loss comparison between the baseline and vectoring cases of C-D nozzle. In this project our main investigation is to measure the exit Mach number and overall pressure loss in the system. Downward deflection of divergent portion by 5, 10, 15 and 20 degree yields Exit Mach number 2.1249, 2.1251, 2.1233, and 2.1052 respectively. The compared to baseline the percentage increased in exit Mach number for 5,10,15 and 20degree downward deflection cases are 2.64%, 2.65%,2.56% and 1.69% respectively. By deflecting the divergent section of nozzle with various angles which can change the exit mass flow rate of the air. Which in turn alters the pressure, velocity and Mach number values at nozzle exit. This variation in Mach number and pressure which leads to thrust vectoring. Downward deflection of divergent portion by 5, 10, 15 and 20 degree gives overall pressure loss in the system 27.30%, 27.58%, 27.49%, and 28.81% respectively. The compared to baseline the percentage increase in overall pressure loss for 5,10,15 and 20 degree downward deflection cases are 14.47%, 14.76%,14.67% and 15.98% respectively. Vectoring cases nozzle gives raises to exit Mach number and also there is an increase in total pressure loss for the given system.

V. CONCLUSION

Whenever demands maneuvering, thechoice is thrust vectoring control system, because they provide excellent survivability and improved performance in the case of gas turbines. The baseline case i.e. C-D nozzle with non-vectoring using S-A turbulence model with 5% Turbulence Intensity resulted into less pressure drop i.e. 12.83 % as compared to k-epsilon and k-omega models.Non-vectoring has attained maximum supersonic flow with exit Mach number using S-A model with 5% Turbulence Intensity is 2.07.Further using S-A turbulence model CFD analysis has been done on C-D nozzle 5, 10, 15 and 20 degree different angles i.e. downward defection of divergent portion of nozzle. It has been found that C-D nozzle gives to increase Mach number. So, thrust vectoring applicable in C-D nozzle. Further deflecting with various angle thrust vectoring will be varying.In downward defection of thrust vectoring nozzle, 10 degree gives maximum increased in exit Mach number i.e.2.1251 which is 2.65% more than baseline case (non-vectoring nozzle) and minimum exit Mach number among vectoring case is given by 20 degree i.e.2.1052 but still it is 1.7% greater than baseline case (non-vectoring nozzle). The appreciable amount of increased in the exit Mach number is seen in vectoring cases compared to non-vectoring case of C-D nozzle; Although increased in pressure loss but for military aircraft key thing is to attend maximum supersonic flow i.e. Mach number. Although difference in exit Mach number among vectoring cases of nozzle are less but it could give qualitative insight.

ACKNOWLEDGMENT

The authors would like to thank the IIAEM Jain university aerospace department giving the opportunity and thankful to Aswatha Narayan guiding me in this research work.

REFERENCES

[1] CFD Analysis of mechanical thrust vectoring control system of missile Umesh Kulkarni1, Ravindra Shende2, Prof. A. K. Gangrade3 2Zeus Numerix Pvt. Ltd., Pune, India 1,3K. J. Somaiya College of Engineering, Mumbai. International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE) Vol 2, Issue 7, July 2017.

[2] Jet Aircraft Propulsion Prof. Bhaskar Roy, Prof. A M Pradeep, Department of Aerospace, IIT Bombay. Lecture No. # 01, Intro and Development of Jet Aircraft Propulsion.

[3] Computational study of mechanical thrust vectoring 1k. Nehru 2t. umarasan. International journal of research in aeronautical and mechanical engineering Vol.3 Issue 10, October 2015.

[4] Aerodynamic Thrust Vectoring for Attitude Control of a Vertically Thrusting Jet Engine by M. Ryan Schaefermeyer Utah State University. Logan, Utah 2011.

[5] Thrust vectoring nozzle for military aircraft engines Daniel Ikaza.Industria de Turbo Propulsores S.A. (ITP), Parque Tecnológico, edificios 300.48170 Zamudio, Spain. ICAS 2000 CONGRESS.

[6] Fluidic thrust vectoring of low observable aircraft Mark S. Mason, William J. Crowther, University of Manchester, School of Engineering, Oxford Road, Manchester M15 9PL. CEAS Aerospace Aerodynamic Research Conference, 10-12 June 2002, Cambridge, UK.

[7] Computational investigation of the aerodynamic effects on fluidic thrust vectoring. Karen Deere, NASA, Langley Research Center, Hampton, VA 36th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference & Exhibit, July 17-19, 2000 / Huntsville, AL