Numerical Analysis of Bypass Mass Injection on Thrust Vectoring of Supersonic Nozzle

Md Shafiqul Islam¹, Md. Arafat Hasan¹ and A.B.M. Toufique Hasan^{1, a}

¹*Department of Mechanical Engineering, Bangladesh University of Engineering and Technology (BUET). Dhaka-1000, Bangladesh*

Abstract. High speed aerospace applications require rapid control of thrust (i.e. thrust vectoring) in order to achieve better manoeuvrability. Among the existing technologies, shock vector control is one of the efficient ways to achieve thrust vectoring. In the present study, bypass mass injection (passive control) was used to generate shock vectoring in a planar supersonic Converging-Diverging (CD) nozzle. Two different bypass lines were used to inject mass in the diverging section varying their dimension in the span wise direction (10 mm × 10 mm² square channel and 2.68 mm × 38 mm² rectangular channel) in such a way that, the mass flow ratio in both the case remain the same (4.9%) in order to compare the effect of bypass channel dimension in the resulting thrust vector angle and thrust performance. Reynolds-averaged Navier-Stokes (RANS) equations with k- ω SST turbulence model have been implemented through numerical computations to capture the three-dimensional steady characteristics of the flow field. Results showed a significant change in the shock structure with the formation of recirculation zone near the bypass injection port in both the case with a variation of shock structure and thrust performance for different geometry bypass lines. It was found that, thrust vector angle increases as injection length increases in the span wise direction.

1 Introduction

Shock vector control (SVC) is one of the fluidic thrust vectoring (FTV) methods and utilizes the shock wave to control the thrust direction. This method using a bypass line was researched in this study. The geometry of nozzle and the experimental data for validation was taken from the Hunter [1] experiment carried out at NASA Langley research center. Two distinctive flow separation phenomena, the free shock and restricted-shock separation, which were observed in experiments with nozzles by Manuel Fray [2] was found on different walls simultaneously and separately in different spans of the studied nozzle. Investigation of Hadjadj [3] gives some key insight on Mach disk, triple point, slip line found on shock structures of separated nozzle flow. In the numerical simulation carried out by Nasuti and Onofri [4] provide an analysis of Mach reflection of the separation shock in over expanded nozzle. Studies of Mittal [5] found interesting phenomenon that the thrust vectoring automatically occurs at low NPR in modified Hunter [1] nozzle. To achieve thrust vectoring Waithe and Deere [6] adapted mass injection by multiple and single port at different NPR to compare their effectiveness. Lately Deng et al.[7] conducted Large Eddy Simulation to achieve thrust vectoring in an asymmetric nozzle

^a Corresponding author: toufiquehasan@me.buet.ac.bd

using bypass mass injection and showed that effective thrust vectoring can be achieved using bypass mass injection keeping the mass flow ratio (ratio of mass flowing through the bypass line to the total mass flow) below 7%. In the present study thrust vectoring was achieved using bypass mass injection on a planar nozzle.

Figure 1. Computational domain with 10×10 sq mm bypass line.

2 Computational Tool

Reynolds-averaged Navier-Stokes (RANS) equations with k- ω SST turbulence model have been implemented through numerical computations to capture the three-dimensional steady characteristics of the flow field. Steady state computaion was performed as suggested by Hunter [1] that the flow structure in the nozzle assumes steday state after NPR 2. The commercial computational fluid dynamics (CFD) code ANSYS Fluent (version 14.5) was used to simulate the flow within the nozzle and the exhaust plume near the nozzle exit using density based solver.

3 Computational Domain

Half of the nozzle was modelled to reduce number of nodes thus computational power and the data obtained was mirrored along the symmetry to get the full flow field inside the nozzle. Computational domain was modeled keeping a 27.483 mm horizontal extra domain prior to inlet and 1258.63 mm horizontal extra domain after the exit to capture the flow structure beyond the exit region of the main nozzle. Grid independency test was done using two dimensional model. Figure 2(b) shows the y^+ value of different mesh resolution, in this study medium mesh with near wall treatment of figure 2(b) having 20000 node in two dimensional model was taken as the base and in three dimension 35 grids was given in the 50.673 mm extrusion in Z direction. The final three dimensional structured mesh contained 1.6 million hexahedral cells and the first length normal to the wall was about $y^+ = 0.4 \mu\text{m}$ near both for the nozzle wall and nozzle side wall to catch the boundary layer properties accurately. Figure. 1 shows the computational domain having 10×10 sq mm bypass line with the boundary conditions.

4 Boundary Condition

The bypass line is so designed that the mass flow ratio remains below 7% to avoid reduction in the resultant thrust which was found in a study by Deng et. al [7]. In the present study with bypass lines of $10 \times 10 \text{ mm}^2$ and $2.68 \times 38 \text{ sq mm}^2$, mass flow ratio is 4.9% was found at NPR 2.4 which is below the maximum percentage of mass flow ratio as suggested by Deng et al.[7]. The operating condition of this study is 0 pa with inlet outlet pressure equal to 2.4×10^5 and 1×10^5 pa respectively. Upper and lower

nozzle wall and side nozzle was as well as the bypass line walls were considered as wall while running simulations. The temperature in all the conditions was fixed to 298.15 K so there was no heat flux through any wall. The pressure far field was assigned with pressure equal to $1e5$ at Mach number 0.1. Turbulent intensity and viscosity ratio at inlet is .1% and 1 respectively whereas at outlet is 5% and 10 respectively. Air was used as the working fluid whose density is assumed vary according to the ideal gas law and its viscosity are assumed vary according to Sutherland's law. Specific heat capacity and thermal conductivity of the fluid are kept constant for simplicity.

Figure 2. Pressure distribution at nozzle wall (a) and wall y^+ at different mesh resolution (b).

5 Validation

Numerical results found from the simulations in the original nozzle at different NPR was plotted in the same graph with the experimental results conducted by Hunter [1] and a complete match in the pressure distribution along the nozzle wall is shown in figure 2(a). After validating the main nozzle, nozzle with bypass line was drawn and meshed following the earlier procedure giving special attention to the bypass channel meshing to ensure better result. Pressure distribution and shock structure is also similar to the study carried out by Hasan [8] using RANS equations with k-omega SST turbulence model in the same nozzle geometry.

6 Analysis of $10 \times 10 \text{ mm}^2$ bypass line (20% span wise)

6.1 Shock Structure.

All the results presented here are computed at NPR 2.4. Shock structure drastically changes due to bypass mass flow injection. The arrow in all the figure shows the mass flow direction in the flow field where x_t is the abscissa of the throat. Lambda shock structure is still present but location of Mach stem has been shifted towards the throat to adjust the incoming flow. So, the flow becomes asymmetric and thrust vectoring is achieved at NPR 2.4. Within 0 to 30mm there is restricted shock separation (RSS) on lower wall and free shock separation (FSS) on upper wall evident from the shock structure presented in figure 3(a). Symmetric shock structure is observed at 31 mm from the symmetry plane shown in figure 3(b). At this injection any slice far from 31 mm from symmetry plane shock structure is reversed. That is FSS on lower wall and RSS on upper wall shown in figure 3(c). As flow approaches nozzle side wall shock gradually diffuses and eventually fades away.

6.2 Streamlines

At the symmetry plane it is clearly seen that, after the bypass mass injection in the diverging sections, streamlines are tilted towards the lower nozzle wall hence the flow is separated from the upper wall. There are 2 pairs of recirculation zone, each pair consisting a clockwise and an anticlockwise vortex,

which are located on the opposite side of the injection port shown in figure 3 (d). Another anticlockwise recirculation zone is also evident at the lower nozzle wall at the end of diverging section. But after the edge of the bypass line, the flow direction gradually changes and reattaches to the upper wall, still reverse flow is observed at the nozzle lower wall at the end of the diverging section. The flow becomes almost symmetric at 31 mm from symmetry plane apart from a small recirculation at the lower wall shown in figure 3 (e). Up to this the main bulk flow is slightly downward but after that main bulk flow is deflected slightly in the upward direction. At 45 mm from symmetry the bulk flow is heavily tilted upward shown in figure 3(f). As moved near the nozzle side wall, streamline direction becomes unpredictable due to the boundary layer interaction of the fluid particle with the nozzle side wall.

Figure 3. Density gradient contour at intermediate slices starting from the symmetry (a), at Z=31 mm (b), Z=45 mm (c), and streamlines at intermediate slices starting from the symmetry (d), at 31 mm (e), and at 45 mm (f).

7 Analysis of 2.68×38 mm² bypass line (75% span wise)

7.1 Shock Structure.

Lambda shock structure is present but location of Mach stem has been shifted towards the throat to adjust the incoming flow. Within 0 to 38 mm (throughout the extension of bypass channel) there is restricted shock separation (RSS) on lower wall and free shock separation (FSS) on upper wall evident from this shock structure presented in figure 4(a) and figure 4(b). Near the nozzle wall complex shock diffusion occurs where shock gradually diffuses and eventually extinguishes at the side wall of the

nozzle as shown in figure 4(c). Throughout the extension of the bypass channel flow pattern is uniform and thrust vectoring is confined within 38 mm from symmetry.

Figure 4. Density gradient contour at intermediate slices starting from the symmetry (a), at $Z=35$ mm (b), at side wall (c) and streamlines at intermediate slices starting from the symmetry (d), at 35 mm (e), at the side wall (f).

7.2 Streamlines

At the symmetry plane it is clearly seen that, after the bypass mass injection in the diverging sections, streamlines are tilted towards the lower nozzle wall. There are 2 pairs of recirculation zone, each pair consisting a clockwise and an anticlockwise vortex, which are located on the opposite side of the injection port shown in figure 4(d). Another anticlockwise recirculation zone is also evident at the lower nozzle wall at the end of diverging section. At 35 mm from the symmetry in figure 4(e), the streamline is almost the same as observed in the symmetry. But after the edge of the bypass line, the flow direction gradually changes and as moved near the nozzle side wall, streamline direction becomes unpredictable due to the boundary layer interaction of the fluid particle with the nozzle side wall which is shown in figure 4(f).

8 Thrust Vector angle and Thrust Performance

The area averaged horizontal and vertical component of velocity at the nozzle exit, mass flow rate and velocity magnitude was calculated from fluent at the nozzle exit. The thrust vector angle is determined

by $\delta = \tan^{-1}(F_y/F_x)$ [2] and thrust is the product of mass flow rate and velocity magnitude at the nozzle exit. A comparison of thrust vector angle and thrust performance is shown in Table 1.

Table 1. Comparison of thrust vector angle and thrust performance

Configuration	Mass Flow Rate (kg/s)	Bypass mass flow ratio (%)	Velocity Magnitude (m/s)	Thrust (N)	X Velocity (m/s)	Y Velocity (m/s)	Thrust Vector Angle
No Bypass	0.812	-----	239.3	194.3	239.3	0.00	-----
10×10 Sq mm Bypass	0.813	4.92	258.27	210	235.59	-14.1413	3.14°
2.63×38 Sq mm Bypass	0.813	4.92	259.34	210.8	237.57	-38.62	9.23°

9 Concluding Remarks

Thrust vectoring using bypass mass injection is a passive mode of thrust control as the injected mass controlling the thrust direction comes from the same combustion chamber producing the thrust. Here in this present study injecting mass by a bypass line at NPR 2.4 on the Mach disc at two different bypass configuration keeping mass flow ratio constant around 4.9% came up with some interesting findings those noted below

- Shock structure changes from symmetric to asymmetric due to bypass mass injection just beneath the injection port for 20 %(10×10 Sq mm bypass) but there is no sign of reverse shock at 75%(2.63×38 Sq mm bypass)
- Due to the presence of reverse shock, integrated effect of thrust vectoring is minimized in 20%
- Recirculation region is observed within the vicinity of injection port and on the nozzle lower wall on both case.
- Flow is span wise more uniform in nature in case of 75% than 20%
- Resultant thrust is vectored by an angle 3.14° for 20% but for 75% it is 9.23°.
- Thrust performance slightly increases from 210 N in case of 20% to 210.8 N in case of 75%.
- It appears that for thrust vectoring 75% bypass will be a better choice than 20%

Acknowledgement

The present work has been carried out with computational resource support from Higher Education Quality Enhancement Project (HEQEP), AIF (3rd Round)-Sub-Project CP 3111, UGC, MoE, GoB.

References

1. C.A.Hunter, *Journal of Propulsion and Power* **20**, 527-532 (2004)
2. M.Frey, *Journal of Propulsion and Power* **16**, 478-484 (2000)
3. A. Hadjadj and M. Onofri, *Shock Waves* **19**, 163–169 (2009)
4. F.Nasuti and M. Onofri, *Shock Waves* **19**,229-237(2009)
5. V.M.K. Kottedda and S. Mittal, *Shock Waves* **27**, 441–455(2017)
6. K. A. Waithe and K. A. Deere, “Experimental and Computational Investigation of Multiple Injection Ports in a Convergent-Divergent Nozzle for Fluidic Thrust Vectoring. AIAA paper. 2003 Jun 23;**3802**:2003.
7. R. Deng, T. Setoguchi, H.D. Kim, *International Journal of Heat and Fluid Flow* **62**, 474-481 (2016)
8. Hasan, ABM Toufique, *International Journal of Heat and Fluid Flow* 46 70-83(2014)