Numerical Investigation and Parametric Study of Fluidic Thrust Vectoring by Shock Vector Control Method

Ananthesha1, S.N. Sridhara2, D. Vamsidhar3
1- Student, M. Sc (Engg), 2- Prof. and Head, Dept of Mech and AE, MSRAS, 3-Sr. Lecturer (AE) M. S. Ramaiah School of Advanced Studies, Bangalore

Abstract
Thrust vectoring is a technique whereby the orientation of the primary exhaust jet from a propulsive unit is varied in order to provide useful aircraft control moments. Thrust vectoring can be achieved, either by mechanical means or by fluidic means. Shock vector control, is one of the fluidic thrust vectoring methods, where in a secondary fluid is injected in the primary flow to deflect the primary exhaust jet.

A numerical study of a axisymmetric nozzle had been carried out, in both 2-D computation and 3-D computation, to determine the angle of deflection of the primary exhaust jet due to the secondary injection. A structured grid of quadrilateral and hexahedral elements was generated in 2-D and 3-D using Gambit. Commercially available computational fluid dynamics code FLUENT was used for computational study. The geometric variables investigated were secondary slot width and location of slot, in the divergent portion. All simulations were computed with a nozzle pressure ratio of 2.42 and a fluidic injection flow rate up to 6% of the primary flow rate.

Results of the numerical study indicate that the shifting of the location of the secondary injection port, in the divergent portion, towards the exit of the nozzle resulted in increased the greater deflection of the primary jet. The increase of the injection slot width, reduced the deflection of the primary jet, for the same mass flow rate in the secondary part of injection. In all 8 secondary configurations. The maximum thrust vector angle achieved was 10° in 3-D as compared to 8.5° in 2-D computation.

Key Words: Fluidic Thrust Vectoring, Shock Vector Control, Secondary Injection, Deflection Angle

Nomenclature

\[ C_{d,\text{skin}} \] Discharge coefficient of primary nozzle, \((W_s + W_p)/W_i\)

\[ W_s \] Measured mass flow rate of secondary nozzle, kg/s

\[ W_p \] Measured mass flow rate of primary nozzle, kg/s

\[ W_i \] Ideal mass flow rate of primary nozzle, kg/s

\[ \delta_v \] Thrust vector or deflection angle, \(\tan^{-1}(V_p/V_s), \text{Deg}\)

\[ V_y \] Velocity of primary jet in y-direction m/s

\[ V_x \] Velocity of primary jet in x-direction m/s

\[ \eta \] Thrust vectoring efficiency, \(\delta_v/(100 W_s/(W_s+W_p)), \text{Deg/}\%

\[ \rho_{i}/(\rho_{i}/(W_s+W_p)) \] Pressure ratio

\[ P_s \] Total pressure, Pa

\[ x/L \] Axial distance ratio

\[ L \] Length of the CD nozzle, m

1. INTRODUCTION

The gas turbine engine does not start and end with the turbomachinery; the inlet and nozzle are essential parts of the engine operation. For military engines, nozzle may be required not only to satisfy observability, but also to be variable in order to handle supersonic flight, afterburner operation or to vector thrust. Thrust vectoring is a technique whereby the orientation of the primary exhaust jet from a propulsive unit is varied in order to provide useful aircraft control moments. Thrust vectoring of aircraft is emerging as a key technology for current and future air vehicles. There are two methods of achieving thrust vectoring - either Mechanically or Fluidic control [1, 2].

Mechanical thrust vectoring involves deflecting the engine nozzle and thus physically changing the direction of the primary jet. These systems are effective, can be heavy, complex, difficult to integrate and aerodynamically inefficient and they require actuated hardware to force the exhaust flow off axis or to vary throat area [1, 2].

Fluidic thrust vectoring involves injecting secondary fluid into or removing it from the boundary layer of a primary jet to enable vectoring. A fluidic thrust vectoring system has the advantage of being lightweight, simple, inexpensive and free from moving parts (fixed geometry), and can be potentially implemented with minimal aircraft observability penalty [1, 2].

Fluidic injection for thrust vector angle control and thrust area control in exhaust nozzles has been studied for over 10 years. Unlike mechanical thrust vectoring nozzles that use actuated hardware to vector the primary jet thrust, fluidic thrust vectoring nozzles use a secondary air stream to manipulate the primary jet flow. Therefore, fixed geometry, fluidic thrust vectoring nozzles have better stealth characteristics and weigh less than their mechanical thrust vectoring nozzles.

The shock vector control method (shown in fig 1(a)) is a fluidic thrust vectoring method, which introduces fluidic injection into the supersonic flow downstream of the nozzle throat. Substantially thrust vectoring efficiencies are generated at the expense of the system thrust ratio as the flow is robustly turned through oblique shocks in the nozzle [2, 3].

Injecting the secondary fluid in the direction of the primary nozzle through secondary inlet port so as to have both the streams flow parallel (co-flow shown in fig 1(b)) and mixing after traveling a considerable axial distance. The secondary jet will have a high velocity creating the low-pressure region. The flow deflects towards the low-pressure region resulting in vectoring of the jet [2-9]. Thrust-shifting methods typically achieve higher system thrust ratios than shock vector control methods, which operate best at off design conditions and have losses due to shocks in the nozzle. Although thrust shifting methods are currently improving, most thrust vector angles reported to date requires up to 6% engine bleed to obtain reasonable amounts of thrust vectoring [2, 3, 10].

The counterflow method (suction in a secondary dust near throat) shown in fig 1(c) provides large thrust vector angles with little secondary flow requirements, but issues such as suction supply source, hysteresis effects, and airframe integration need to be addressed [2, 3].

In the first part of the current study, numerical simulation of Shock Thrust Vectoring (STV) in 2-D axisymmetric supersonic nozzle is carried out to understand the physics of thrust vectoring. Further, the parametric studies on the STV which include the variation of secondary injection slot location and variation of slot width, in the divergent portion of the CD nozzle using both 2-D and 3-D computation.

2.1 Governing Equation

The computational fluid dynamics code FLUENT solves the three-dimensional, Reynolds-averaged Navier-Stokes (RANS) equations and use one of the turbulence model for closure of the RANS equation. The governing equations, which include the conservation equations for mass, momentum, and

Vol. VI, No.2, September 2007
2.3 Boundary Conditions

The FLOW3D code has many options for defining the conditions of the inflow, outflow, free-stream, wall, symmetry and centerline boundaries. For this study, Riemann invariants along the characteristics were implemented along the lateral and inflow free stream boundaries. At the downstream boundary, a subsonic, constant pressure outflow boundary condition was used, which automatically extrapolates pressure and all other flow quantities from the interior if the flow is supersonic. The primary nozzle flow was specified with a fixed total-temperature and total-pressure boundary condition and the fluidic injection flow was specified with varied mass flow rate and constant total temperature. A no-slip, adiabatic wall boundary condition was implemented on nozzle surfaces to obtain viscous solutions. The Mach number of 0.01 is specified for the ambient region (far field) to avoid the solution divergence due to large gradient.

2.4 Solver Setting

FLUENT is a commercially available state-of-the-art computer program for modeling fluid flow and heat transfer in complex geometries, which uses a control-volume-based technique to convert the governing equations to algebraic equations, that can be solved numerically to obtain fluid velocities, pressure, temperature and other fluid properties [4].

Using segregated approach, the governing equations are solved sequentially (i.e., segregated from one another). Because the governing equations are non-linear (and coupled), several iterations of the solution loops are performed before a converged solution is obtained to solve these algebraic equations.

2.5 Nozzle Geometry

An axisymmetric nozzle with a throat area of 18.5 m² and an expansion ratio (exit area divided by throat area) of 1.03 was modeled. The geometric configuration is shown in fig. 2. The inlet pressure and temperature for primary nozzle are 2.45 bar and 829.3K respectively.

A secondary fluid at 725K is injected into the divergent portion of the primary nozzle with a varied mass flow rate from 0.35% to 6% of the primary flow rate.

Two slot positions, 400 mm and 440 mm in the divergent portion of the CD nozzles located towards the nozzle exit and measured from the nozzle inlet were used. The injector slot widths of 18 mm, 22 mm, 28 mm and 35 mm were considered for the 2D and 3D computations. The slot injector is assumed to have a 1 m² depth in third dimension in 2-D analysis. In 3-D computation the secondary injection slot angle was taken as 60(30°) = 0 along the circumference of the divergent portion of the nozzle.

Fig 2. Primary nozzle specifications

2.1 Computational Domain

Two-dimensional mesh was used in order to quickly evaluate a multitude of geometric parameters. Quadrilateral and hexahedral elements are used to mesh the 2-D and 3-D (shown in fig 3 and fig 4) fluid domain respectively.

Vol. VI, No. 2, September 2007
The shock waves from the internal flow of the nozzle should not interact with specified boundary conditions. If the shocks are interacting with the specified boundary conditions, the numerical instability will occur which causes the solution to diverge. The free stream domain was located 3-throat diameter upstream and 25-throat diameter downstream of the nozzle exit. The upper and lower lateral free stream domain were located 20-throat diameter above and below the nozzle axis based on the open literature [3, 5, 6].

If the boundary layer is turbulent, it appears to reflect the shock waves as a shock wave in much the same way as would the solid surface in the absence of the boundary layer, would do with some shear thickening. There may also be local separation and reattachment, in which case the reflected shock originates just ahead of the point of incidence [5]. To eliminate the reflected shocks the boundary layer was defined for a law-of-the-wall coordinate of $y^+ = 0.5$ on the fine mesh spacing for adequate modelling of the boundary layer [7, 8, 9].

![Two-dimensional mesh domain](image1)

**Fig 3. Two-dimensional mesh domain**

![Three-dimensional mesh domain of nozzle and inlet flow domain](image2)

**Fig 4. Three-dimensional mesh domain of nozzle and inlet flow domain**

75,460 quadrilateral elements in 2D and 9,88,880 hexahedral elements in 3D were used for the study of secondary injection in the fluidic thrust vectoring.

RESULTS AND DISCUSSIONS

3.1 Validation Study

Dual throat thrust vectoring nozzle concept [5] was simulated as a part of validation with experimental results reported by Flam et al [8]. Experimental data acquired to date on the dual throat nozzle has been at static free stream conditions. Therefore, computational solutions were simulated with a free stream, although a small convective Mach number of $M = 0.01$ was used for computational stability.

Simulations of dual throat nozzle configurations operating at NPRs 2, 4 and 7 were computed with no injection and 1 to 4% injection. The comparison of computationally predicted results with experimental data for discharge coefficient without secondary injection is shown in fig 5.

![Comparison of experimental and computational primary discharge coefficient with no secondary injection](image3)

**Fig 5. Comparison of experimental and computational primary discharge coefficient with no secondary injection.**

![Comparison of experimental and computational prediction of thrust deflection angle for NPR = 4.0](image4)

**Fig 6. Comparison of experimental and computational prediction of thrust deflection angle for NPR = 4.0.**

While there is generally good agreement between the predictions and experimental data, the computational $C_{p,w}^{prim}$ results are higher than the experimental $C_{p,w}^{prim}$ results over the NPR range tested. Higher discharge coefficient levels for the computational results can be attributed to vena contracta effects, differences between the computational mesh or turbulence model and the as-built test article [5].

A discharge coefficient less than one indicates that the actual flow rate is less than the ideal flow rate for a given area. The vena contracta effect occurs when the flow along the walls at the upstream throat cannot manipulate the turn and overshoots the angle resulting in an effective area smaller than the actual geometric area. The geometric minimum area is larger than the area of high total pressure flow restricted through the vena contracta effect.
Fig 7. Comparison of experimental and computational prediction of thrust vectoring efficiency for NPR = 4.0.

Fig 8. Surface pressure distribution of Dual throat Nozzle at (a) Lower wall (b) Upper wall at NPR = 4 with 3% injection

Thus, the reduced actual flow area from the vena contracta effect caused lower discharge coefficients compared to the discharge coefficient obtained without vena contracta effect was absent from the flow.

Fig 9. Experimental shadowgraph of dual throat nozzle. NPR=4, 3% secondary injection [8].

Fig 10. Computational Mach contours using FLUENT of dual throat nozzle. NPR=4, 3% secondary injection.

Variation of thrust deflection angle and thrust vectoring efficiency with an increase in secondary mass flow rate are shown in fig 6 and 7 respectively. Thrust vectoring angle was predicted within less than 1, while discharge coefficient and system thrust vector efficiency predictions fell within 1% of experimental results. Two main differences between experimental data and computational predictions are the absence of the viscous sidewalls for all computational simulations and the absence of the plenum and injection opening in the no injection simulations. The later explains the 0.8 discrepancy in thrust vector angle, with the computational prediction of 0.

Fig 8(a) and (b) shows a comparison of the 2-D quasi slot computational prediction with pressure data for the 3-D experimental slot configuration for NPR = 4 with a 3% injection rate. The most obvious difference between experimental and CFD is the flow expansion and shock location on the upper wall, and the pressure along the lower wall are pumped down to a lower pressure with CFD result. FLUENT did a fairly good job predicting the flow characteristics of the 3-D slot injection along the upper wall, although the shock was slightly stronger and not in the exact location of the experiment. FLUENT was not able to predict the low pressures in the separated flow regions along the lower wall. The curve is flat where the flow is separated along the lower cavity wall, and there is a slight increase in the pressure near the nozzle exit as the primary jet vectors down and impinges on the lower surface.
Fig 11. Pressure distribution in two-dimensional at (a) Lower wall, (b) Upper wall.

Fig 9 and 10 represent the experimental shadowgraph image and Mach contours obtained computationally, for NPR=4, 3% secondary flow injection. The computational Mach contours capture the qualitative flow features seen in the shadowgraph image. Both image shows the large separated flow region in the lower cavity downstream of the secondary flow injection and the small separated flow region in the upper. The normal shock and lambda foot downstream of the throat are also captured by computationally.

The computational results predicted $\gamma = 11.85$, $\gamma = 3.36$ per percent secondary injection, $C_{\text{comp}} = 0.884$. The experimental data results for this case were $\gamma = 11.7$, $\gamma = 4.0$ per percent secondary injection, $C_{\text{comp}} = 0.877$. There is a good agreement between the computationally predicted and measured thrust vector angle but a deviation of 1 per percentage in $C_{\text{comp}}$. The computational results in predicting $C_{\text{comp}}$ is again optimistic.

3.1 Results

Figure 11 shows the pressure distribution across lower wall and upper wall for injection slot width of 22 mm at locations $x/L = 0.8$ and $0.9$. The pressure reduces through the convergent portion till the throat region and increases in the divergent section indicating the functioning of a nozzle. It is observed that the low-pressure region obtained due to secondary injection at $x/L = 0.8$ is higher compared to $x/L = 0.9$. Hence the vector angle obtained with the injection at $x/L = 0.8$ is lower compared with injection at $x/L = 0.9$. It was observed that the same pressure distribution trend for lower and upper wall region shown in Fig. 12 with an increase in the secondary injection mass flow rate from 0.35% to 6% of the primary mass flow.

Figure 12 shows the pressure distribution across lower wall and upper wall for injection slot width of 22 mm at locations $x/L = 0.8$ and $0.9$ for 3-D computation. The pressure reduces through the convergent portion till the throat region and increases in the divergent section indicating the functioning of a nozzle. It is observed that the low-pressure region obtained due to secondary injection at $x/L = 0.8$ is higher compared to $x/L = 0.9$. Hence the deflection obtained with the injection at $x/L = 0.8$ is lower compared with injection at $x/L = 0.9$. It was observed that the same pressure distribution trend for lower and upper wall region shown in Fig. 12 with an increase in the secondary injection mass flow rate from 0.35% to 6% of the primary mass flow. The results of 2-D and 3-D computations were in good agreement.

SASTECH

Vol. VI, No.2, September 2007
3.2.1 Thrust deflection angle

To study the effect of slot position and slot location on thrust vector angle was carried out using both 2-D and 3-D computation.

Effect of slot position

The graph in Fig. 13 shows the variation of the thrust vector angle versus the ratio of secondary mass flow rate \(W_s\) to the sum of primary mass flow rate \(W_p\) and secondary mass flow rate \(W_s\), i.e., \(W_s/W_{p+s}\) for 2-D computation.

With the shift of injection location towards the nozzle exit in the divergent portion, increase in the thrust vectoring deflection angle was observed. Due to the decreased low-pressure region with injection location at \(x/L = 0.9\) compared to with injection location at \(x/L = 0.8\) the thrust vectoring deflection angle is greater; in the case with injection location at \(x/L = 0.9\). With the increase of the secondary injection mass flow rate also, there is an increase of the thrust vectoring deflection angle, which is due to increased momentum.

Figure 14 shows the 3-D computation thrust vector angle for different secondary injection locations and varied slot widths for different secondary mass flow rate varying from 0.35% to 6% of the primary mass flow rate.

Effect of slot width

Figure 13 and 14 show the variation of the thrust vectoring deflection angle with the variation of the secondary slot width, in 2-D and 3-D computation respectively. Lower width slot gives the higher thrust vectoring angle compared to other secondary slot width configurations for the same mass flow rate of injection and for the same position. This is due to the fact that the momentum in the lower width secondary slot is more as compared to other secondary slot width configurations.

From Fig. 15, it is evident that for the same width of the port (22 mm), in both 2-D and 3-D computation the deflection of the primary jet obtained from the 3-D computation is lower as compared to that obtained form 2-D computation, due to the assumption of 1 m depth of the slot in the 2-D computation, but in 3-D computation the slot is modeled as 60 from the center along the circumference of the nozzle, which produces more momentum, to deflect the primary jet. At lower secondary injection rate all the configuration are showing the same vector angle within 0.4 for all configurations. The greater angle from 3-D computation is 10° whereas for 2-D computation it is 8.5°.
3.2.1 Thrust vectoring Efficiency

Figures 16 and 17 show the variation of thrust vectoring efficiency versus the ratio of secondary mass flow rate (Wp) to the sum of primary mass flow rate (W) and secondary mass flow rate (Wp), i.e., W/(Wp + W) obtained from both 2-D and 3-D computation. Thrust vectoring efficiency [10] obtained from 3-D computation is lower for lower secondary mass flow rates as compared to that obtained from 2-D computation which is due to the fact that the jet obtained from 2-D computation is having a greater deflection angle compared with 3-D computation for lower secondary mass flow rates. But a reverse trend is noticed when higher secondary injection mass flow rates are used, the thrust vectoring efficiency as obtained from the 3-D computation is greater as compared to that obtained from 2-D computation, due to the greater deflection angle obtained from the jet in the 3-D computation for the same flow rate as compared to that obtained in 2-D computation which is evident from Fig. 17 and 18.

Fig 17. Thrust vectoring efficiency in three-dimensional

Fig 18. Computational Mach number contours in 2-D (a) without secondary injection (b) with 4% injection.

Fig 19. Computational Mach number contours in 3-D (a) without secondary injection (b) with 4% injection.

The fig 18 (a) and (b) gives a qualitative image of Mach number contours in 2-D computation without and with secondary mass flow injection. Fig 19 (a) and 19 (b) gives a qualitative image of Mach numbers contours at the centre-plane in 3-D computation without and with secondary mass flow injection. The shocks are disappearing after the injection in the fig 18(b) as compare to 18(a) in 2-D computation. Similar trend with higher deflection angle is observing in the 3-D computation Mach number contours shown in fig 19 (a) and (b). The smooth and continuous contours of the axisymmetric duct allowed an efficient pressure relief to occurs around the injection slot, thereby reducing the strength of the shock system and the loss in thrust efficiency is also observed in the fig 19 (a) and (b).

CONCLUSIONS

Computational investigation has been carried out to access the use of fluidic injection to control the deflection angle of a nozzle to enhance the thrust vectoring capability of a nozzle with shock vector technique. Secondary port geometric variables included slot width and slot position in the divergent portion. Distinct conclusions from this work include:

- The CFD prediction of thrust vector angle and thrust vectoring efficiency are well within 1 and 1% respectively as compare to experimental result.
- At a given operating conditions, maximum thrust deflection angle was achieved by 10 in 3-D computation as compared to 8.5 in 2-D computation for a secondary injection rate of 0% at x/L = 0.9 and slot width = 35mm.
- The thrust vectoring efficiency exhibits lower losses in the 3-D computation when compared to 2-D computation at similar secondary injection conditions.
- Injection slot width had minor effects on the generation of thrust vector angle in the lower secondary flow rate up to 2.5%. More than 2.5% secondary injection rate, larger variation in the thrust deflection angle is observed.
• Increase in the secondary slot width decreased the thrust vector angle without significant penalty in the thrust vectoring efficiency at the same secondary injection rate.

• At a given primary jet conditions, shifting the injection port towards nozzle exit, increased the thrust vector angle by reducing the lower pressure region in the injection side of the trailing edge.

REFERENCES


SAS TECH

65 Vol. VI, No.2, September 2007