

Numerical investigation of flow deflection due to secondary injection for various pressure ratios in C-D nozzle

C. Suresh¹ G. Dinesh Kumar, D.Gowrishankar

csuresh2u@gmail.com¹

Assistant Professor, School of Aeronautical Sciences,
Hindustan Institute of Technology & Science, Chennai, India

Abstract

Fluidic thrust vectoring provides an additional control variable that offers many benefits in terms of maneuverability and control effectiveness in the new generation aircrafts and missiles. Thrust vectoring capabilities make the satisfaction of take-off and landing requirements easier. Fluidic thrust vectoring continues to work at low dynamic pressures, where other control technologies are less effective, making it a more valuable and dynamic in flight envelope. Additionally, thrust vectoring could increase conventional controls for some control power to trim the aircraft and thus reduce cruise trim drag. In this paper, an attempt has been made to numerically study the effect of thrust vectoring using shock vector method for various pressure ratios (Nozzle inlet pressure to Secondary injection Pressure) and the jet deflection angles are obtained. Also aims to study the effect of nozzle blockage effects for each pressure ratios. The numerical study includes two different inlet pressure conditions namely 9 and 12 bars, with the secondary injection pressure ratio varied from 9 to 15 bars. The numerical study is based on the K-epsilon turbulence model with enhanced wall treatment to capture the complex flow phenomena occurring on upstream and downstream of the secondary injection port. The primary jet decay characteristics at the nozzle exit and the side forces created on the nozzle walls are studied.

Key words: C-D nozzle, Thrust vectoring, Flow deflection, flow separation, fluidic thrust vectoring, shock vector.

Corresponding author, email: csuresh2u@gmail.com,

Introduction

Fluidic thrust vectoring is one of the prime technologies that are needed to maneuver the next generation aircraft. The complexity of mechanical systems and the need to protect the components from harmful conditions induce substantial weight, maintenance, and design constraints. To avoid these major drawbacks, attention has been focused on the development of fixed-geometry thrust vectoring systems that employ fluidic flow control [1,2]. Three primary mechanisms of fluidic thrust vectoring

methods that have been studied over the last few years are shock-vector control, throat shifting, and counter flow [3]. These techniques are used to vector the exhaust flow in the pitch and yaw directions [6]. Fluidics, on the other hand, offer reduced weight, higher reliability, and in the case engine airframe integration they may also be more easily integrated with airframe structures to share loads and eliminates redundant structures [5,6]. Nozzle thrust-vectoring efficiency is defined as the

magnitude of the thrust vector angle in degrees divided by secondary injection flow as a percentage of total jet flow, is an indicator of flow turning efficiency [4].

Computational Domain

The flow in convergent divergent nozzle was simulated numerically using the commercial software Fluent. Both convergent and divergent sections are included and at 80% of the diverging section a secondary injection slot of 2mm diameter is located. A structured mesh containing 20695 nodes was used. Using the isentropic relations, C-D nozzle was designed for the exit Mach number 2.5 and total nozzle length of 160mm, shown in fig.1. To avoid the sharp edges at the nozzle throat a small radius of curvature is provided at the throat.

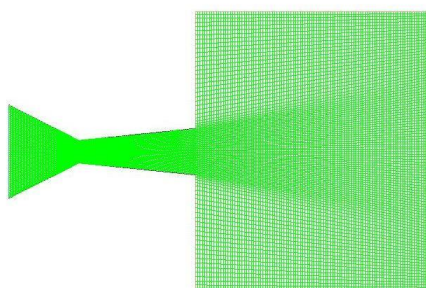


Fig.1 C-D Nozzle with structured mesh

The grid/mesh independence of the solution was tested by varying the number of nodes for three different meshes. The center line velocity variation along the length of the nozzle was compared for all the three different meshes and an excellent agreement was observed shown in fig.2

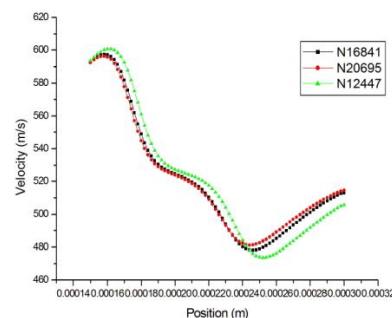


Fig.2 Grid independence test

Numerical Procedure

Two dimensional, density based Navier-Stokes formulation has been used for the current analysis to understand the flow behavior and shocks. Three dimensional analysis will be carried out after the current study. The ideal gas equation has been engaged to predict the density and viscosity variation of the working gas. No-slip wall conditions are used on the nozzle walls. Pressure inlet of 9 and 12 bar are used as boundary conditions were used at the inlet and secondary inlet pressure are varied from 4.5 to 9 bar and 6 to 12 bar corresponding to 9 and 12 bar inlet pressures respectively, Pressure outlet at the exit. The external domain with far field boundary condition was created at the exit of the nozzle to find the jet deflections. Two equations K- ϵ , RNG with enhanced turbulence model was used to solve the Navier-Stokes equations. This model was selected to capture the complex flow phenomena occurring on upstream and downstream of the secondary injection port and due to its computational efficiency for nozzle flows.

Results and discussion

Secondary injection flow in the divergent section of nozzle, results in change of flow direction from the axial to transverse direction. Flow separates upstream of the secondary injection and a recirculation region is formed shown in fig.3. The vortex formed due to this recirculation affects the wall pressure distribution.

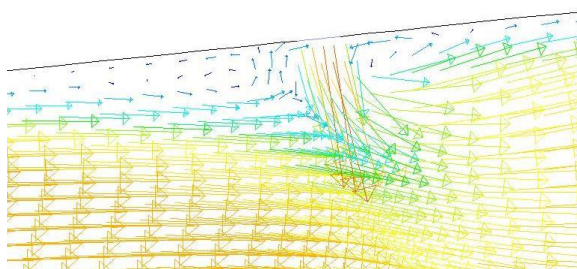


Fig.3 Recirculation of flow at secondary injection

The results of fluidic thrust vectoring for inlet pressure of 9 bar are represented in case1 and for 12 bar inlet pressure in case 2.

Case: 1(Inlet pressure of 9 bar)

The velocity contour plots for pressure ratio of 0.6, 0.8 and 1 are shown in fig 4, 5 and 6. From contour plots we can visualize the increase in vectoring angle with increase in pressure ratio.

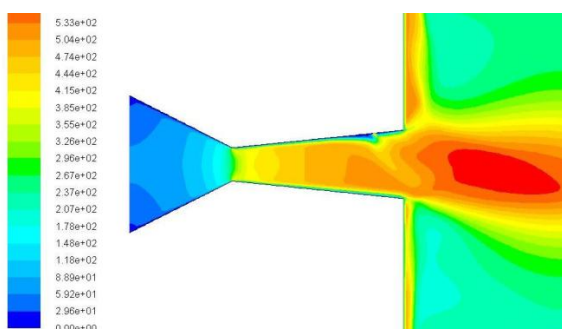


Fig 4 Velocity contour for Pressure ratio 0.6

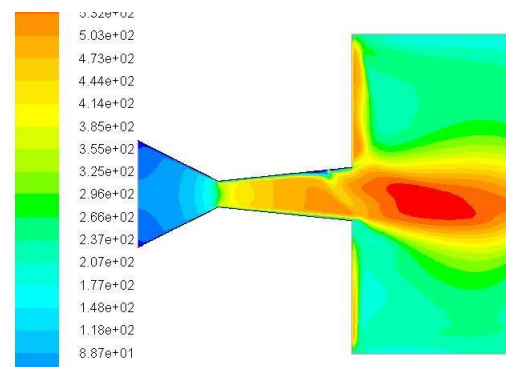


Fig 5 Velocity contour for Pressure ratio 0.8

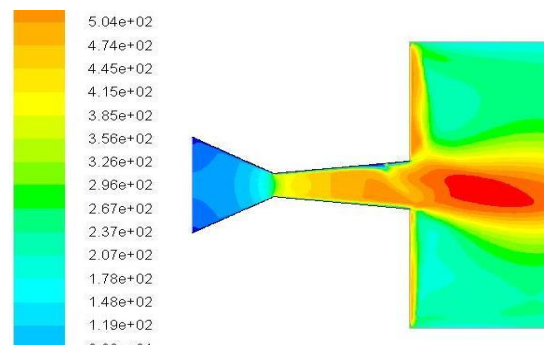


Fig 6 Velocity contour for Pressure ratio 1

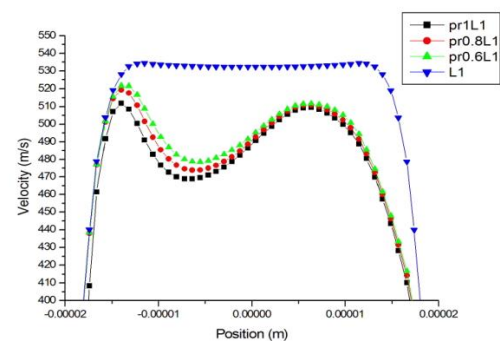


Fig.7 Velocity plot with and without injection at L1

The velocities in the exit plane of nozzle are measured at different locations (at exit of nozzle L1, 50mm downstream of nozzle exit L50 and 100mm downstream of the nozzle exit L100) for all the pressure ratios of 0.6, 0.8 and 1. The velocity plot without injection and with injection for all three pressure ratio is compared

at different location of the exit plane L1, L50 and L100. Fig 7 represents the velocity plot at location L1, the nozzle exit. When secondary fluid is injected, at the nozzle exit plane the velocity distribution is changed from the axial direction. The change in flow direction and the change in velocity magnitude are compared with the flow without secondary injection.

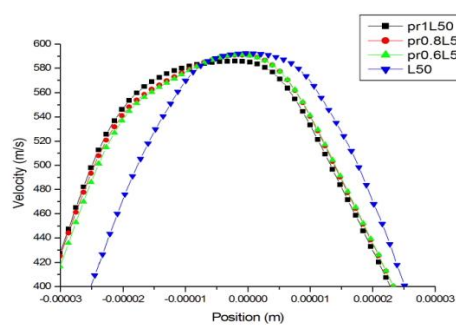


Fig.8 Velocity plot with and without injection at L50

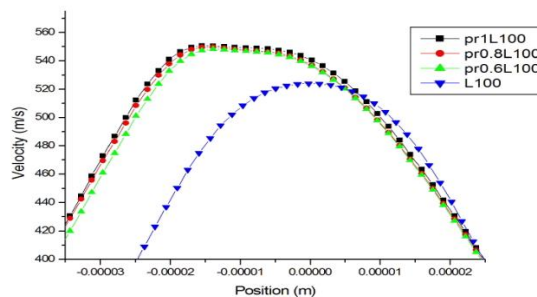


Fig.9 Velocity plot with and without injection at L100

Fig 8 and 9 represents the velocity plot at location 50 mm and 100mm downstream of the nozzle exit. The flow deflection at 50 mm and 100mm from the axial direction increases as we move downstream of the nozzle. In fig 9 it is observed that the velocities of deflected jets are greater than the jet without deflection, Also the deflection angle at 100mm are greater than deflection at 50mm locations.

Case: 2 (Inlet pressure of 12 bar)

The velocity contour plots for pressure ratio of 0.6, 0.8 and 1 are shown in fig 10, 11 and 12.

Fig 10 Velocity contour for Pressure ratio 0.6

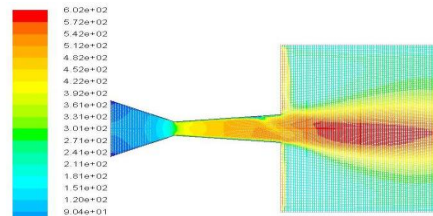


Fig 11 Velocity contour for Pressure ratio 0.8

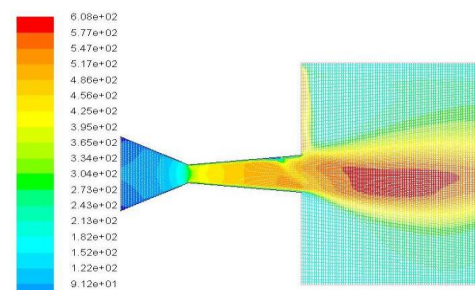


Fig 12 Velocity contour for Pressure ratio 1

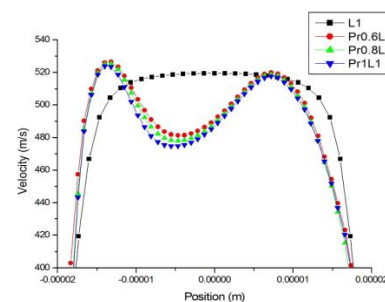


Fig.13 Velocity plot with and without injection at L1

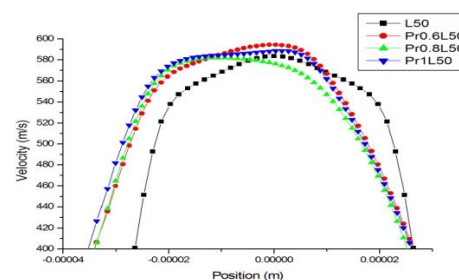


Fig.14 Velocity plot with and without injection at L50

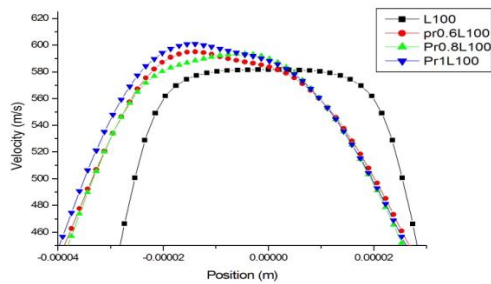


Fig.15 Velocity plot with and without injection at L100

The flow deflection behavior for 12 bar inlet pressure is similar to flows at 9 bar, but the deflection angle increases as the injection pressure increases. In this case at low secondary injection pressure the deflection angle is less compared to case 1, as the primary gas pressure is larger.

Conclusion

A small scale two dimensional convergent divergent nozzle was used to investigate the performance of fluidic thrust vectoring using shock vector control method. A secondary injection port was formed by a slot normal to the diverging wall of the nozzle. The analysis was made for two different inlet pressure ratios of 9 and 12 bar. The velocities downstream of the nozzle are measured to find the flow deflections. It is observed that if the secondary injection pressure is increased above pressure ratio of 1, the flow separation dominates which leads to reverse flow inside the nozzle leading to divergence. A good agreement was observed for both the cases if the secondary injection pressure ratio is less than 1. The turning of the primary flow stream by the shock and the unbalanced pressure increase due to secondary

flow injection, which is induced on the nozzle wall where the boundary layer upstream of the injection separates.

Injection slot on the wall needs to be profiled out of the nozzle and not to impose new boundary condition on the nozzle wall itself which creates numerical artifacts and not physical solutions. This should be either just strait “pipe” section or convergent section exiting into the nozzle. In Fig 4 to 6, the velocity contours mentioned the effects on the convergent and throat sections are taking its toll. Upper corners close to inlet are not properly resolved generating back flow region. Due to short curvature radius at throat, there is strange artifact on the velocity profile propagating downstream and mixing with compression waves that are typical for the convergent nozzle. The main supersonic flow reacts on the obstacle in the shape of secondary injection with the bow shock. This bow shock inflicts the adverse pressure gradient in the upstream zone of the injection, which consequently detaches the flow from the nozzle wall. In the separation zone there are 2 counter rotating vortices in Fig.3 one can denote 2 vortices, primary longer close to incipient point and secondary counter rotating near the injection itself. Finally the two dimensional model is not very suitable for secondary injection thrust vectoring problems

References

1. M. S. Mason and W. J. Crowther, "Fluidic thrust vectoring for low observable air vehicles", 2nd AIAA Flow Control Conference, 28 June -1 July 2004, Portland, Oregon
2. Karen A. Deere, Bobby L. Berrier, Jeffrey D. Flamm, "A Computational Study of a New Dual Throat Fluidic Thrust Vectoring Nozzle Concept", 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit, 10-13 July 2005, Tucson, Arizona.
3. Jeffrey D. Flamm*, Karen A. Deere, Bobby L. Berrier, Stuart K. Johnson, and Mary L. Mason, "Experimental Study of a Dual-Throat Fluidic Thrust-Vectoring Nozzle Concept", 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 10-13 July 2005, Tucson, Arizona
4. Andrew J. Neely, Fernando N. Gesto and John Young, "Performance Studies of Shock Vector Control Fluidic Thrust Vectoring", 43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit 8-11 July 2007, Cincinnati, OH.
5. D. J. Forliti and I. Echavarria Diaz-Guardamino, "Exploring Mechanisms of Fluidic Thrust Vectoring Using a Transverse Jet and Suction", AIAA JOURNAL, Vol. 47, No. 10, October 2009.
6. K.A Deere, "Computational investigation of the aerodynamic effects on Fluidic thrust vectoring", AIAA 2000-3598, NASA, Hampton, VA
7. Zmijanovic V, Leger L, Depussay E, Sellam M, and Chpoun A, "Experimental-Numerical Parametric Investigation of a Rocket Nozzle Secondary Injection Thrust Vectoring", Journal of Propulsion and Power, Vol. 32, No. 1 (2016), pp. 196-213. doi: 10.2514/1.B35721
8. Hilbert van Pelt, Andrew Neely, and John Young, "A System Study On Fluidic Thrust Vectoring", 20th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, 6-9 July 2015, Glasgow, Scotland.

