International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.3., Issue.3, 2015

RESEARCH ARTICLE

ISSN: 2321-7758

DESIGN OPTIMIZATION OF FLUIDIC THRUST VECTORING NOZZLE USING CFD

RAJESH.R[1], ANBUCHEZHIYAN.G[2]

[1]M.E CAD/CAM, Dept of Mechanical Engineering, Valliammai Engineering College, Kanchipuram , Tamilnadu , India.

[2]Assistant Professor, Dept of Mechanical Engineering, Valliammai Engineering College, Kanchipuram , Tamilnadu , India

Article Received: 29/04/2015 | Article Revised on:03/05/2015 | Article Accepted on:05/05/2015

RAJESH.R

ABSTRACT

Thrust vectoring is a technique where from the primary exhaust jet from propulsive unit is useful to provide aircraft control moments. Thrust vectoring can be achieved by either 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 to the primary flow to deflect the primary exhaust jet by which a side force will be generated. Numerical simulation of fluidic thrust vectoring in C-D nozzle will be carried out for various pressure ratios of secondary fluid. The work presented in this paper deals with the development of a shock vector control by fluidic thrust vectoring system for use on a low observable Unmanned Air Vehicle (UAV) operating in Supersonic flight regime. Computational analysis on shock vector control gives positive results up to 18 degree of flow deflections. Further studies are being carried out on the same and results are validated with the existing experimental results.

Keywords—C-D nozzle, Fluidic Thrust vectoring, Shock wave, Supersonic flow.

I. INTRODUCTION

Thrust vectoring technique is employed to augment the ability of a supersonic aircraft to manipulate the direction of the thrust from its engines. Although many studies have been carried out in fluidic thrust vector nozzles the design optimization of thrust vector control (TVC) is still a daunting task in aerospace industry. Thrust vectoring allows the aircraft to perform various maneuvers not available to conventional aircrafts in combat situations. Integration of thrust vectoring nozzles with the aircraft configuration is known to optimize the lift distribution during cruise, resulting in reduced sonic-boom signatures[3]. Thrust vectoring may be employed to have better control at low dynamic pressures when the conventional control technologies become less effective. Supersonic aircrafts employ a variable area nozzle to achieve optimum performance throughout the flight envelope. Two methods of implementing thrust vectoring are fluidic and mechanical.

Fluidic thrust vectoring systems have the advantage of fewer moving parts, resulting in lesser weight compared to mechanical thrust vectoring nozzles. In fluidic thrust vectoring the exhaust flow is manipulated with the use of a secondary air source, usually the bleed-air from the engine compressor or fan. Three primary mechanisms of fluidic thrust vectoring are shock vector control, throat shifting, and counter flow[4]. These techniques can be used to vector the exhaust flow in the pitch or yaw direction.

Computational Fluid Dynamics (CFD) has become an integral part of the engineering design and analysis environment that require the availability to predict performance of new designs or processes before they are ever manufactured or implemented.

II. METHODOLOGY

A. Design work

The work involves designing of fluidic thrust vectoring C-D nozzle for use on a low observable Unmanned Air Vehicle (UAV) operating in Supersonic flight regime. The designs of the nozzle have been made using CATIA V5 software for flow analysis. The typical Fluidic thrust vectoring nozzle consists of a single secondary port for the injection of secondary jet.

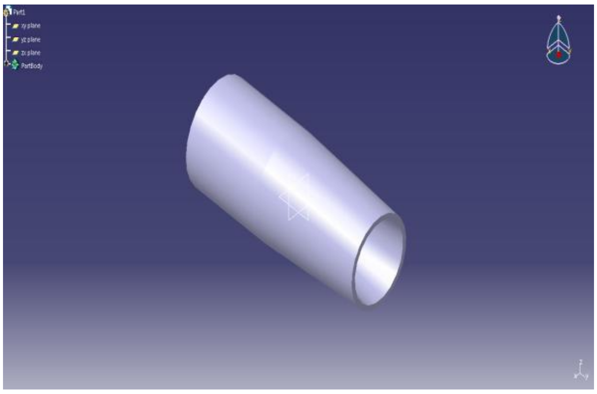


Fig.1. Nozzle with outer covering.

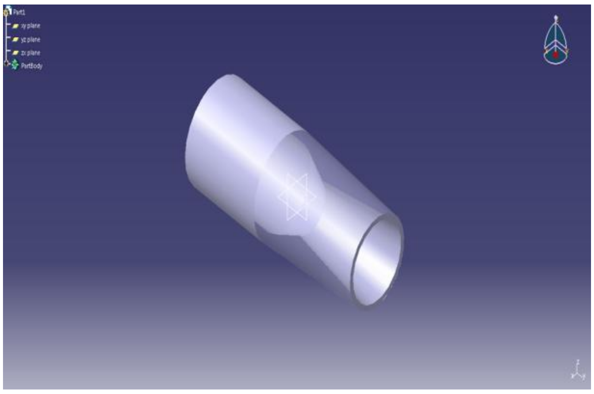


Fig.2. Nozzle cross sectional view.

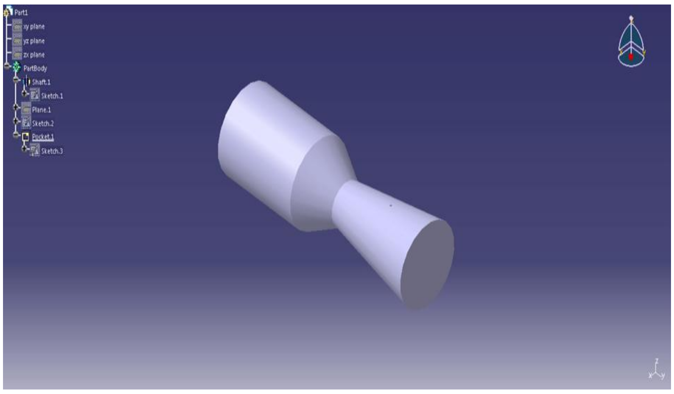


Fig.3. Convergent – Divergent Nozzle.

B. Computational Analysis

The present study investigates the characteristics of gas flow through a FTV Nozzle with varying cross sections. CATIA V5 model was imported in ANSYS CFX 14 preprocessing tool and meshed. In order to capture velocity boundary layer the entire model was discredited using fine mesh elements to get accurate result.

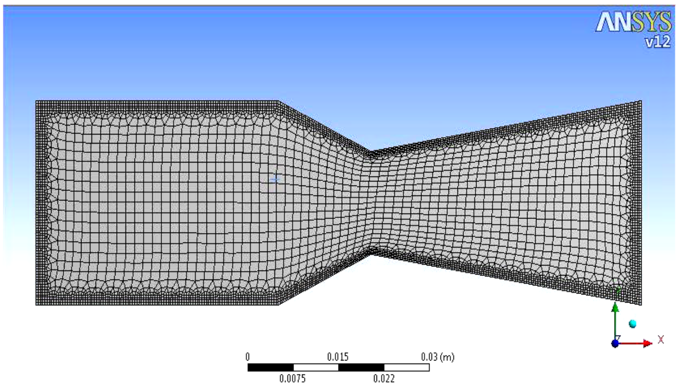


Fig.4. Grid system in C-D nozzle

C. Computational domain and Boundary conditions

The boundary conditions used are Pressure Inlet at the entrance to the nozzle, Pressure Outlet at the outer boundary of the computational domain and adiabatic no-slip conditions at the wall. The secondary blowing nozzle flow is defined by mass flow inlet. The total pressure (Po) and total temperature (To) were applied at the inlet with the value of Po being varied from 1 MPa to 200 kPa and To being maintained at constant value of 293.15 K. At outlet nozzle back pressure (Pb) was assumed to be the same as the ambient pressure.

The nozzle pressure ratio (NPR=Po/Pb) was varied in the range of 2.0 to 10.0 to obtain overexpansion flow at nozzle exit. Blowing of the secondary mass flow rate was varied from 0% to 10% of the total inflow.

Fig.5 describes the computational grid and the boundary conditions employed in the present study.

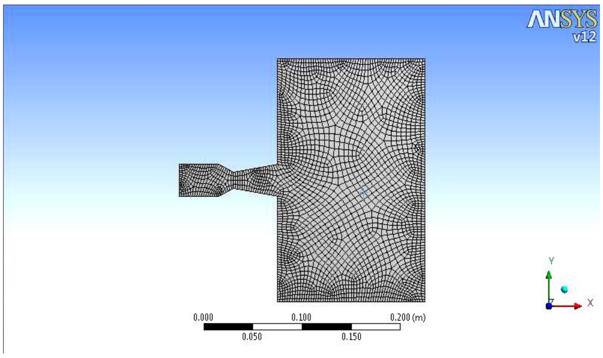


Fig.5. Grid system in C-D nozzle with domain.

RESULT AND DISCUSSION

In this paper the forces produced by the exiting exhaust gases are manipulated from the axial direction to produce a side or vertical force by injecting a secondary jet with different jet pressures from various divergent locations of the primary nozzle to examining the desirable secondary jet characteristics of an efficient TVC system.

Computational analysis has been carried for the nozzle designed with various cross sectional area. The Velocity streamline, Pressure plane for various design cases are predicted using postprocessor tool.

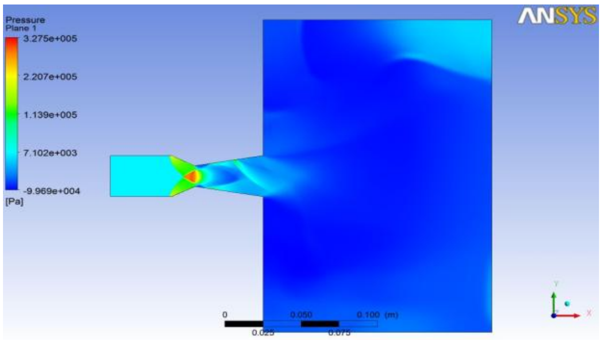


Fig.6. Pressure plane of C-D nozzle.

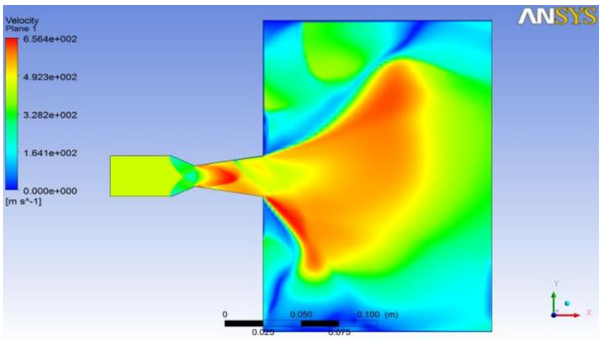


Fig.7. Velocity plane of C-D nozzle.

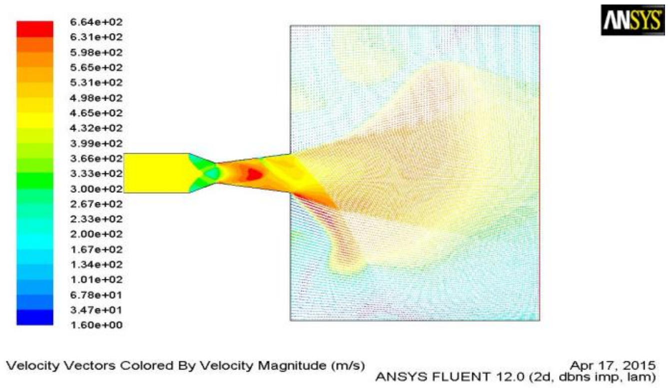


Fig.8. Velocity streamline of C-D nozzle.

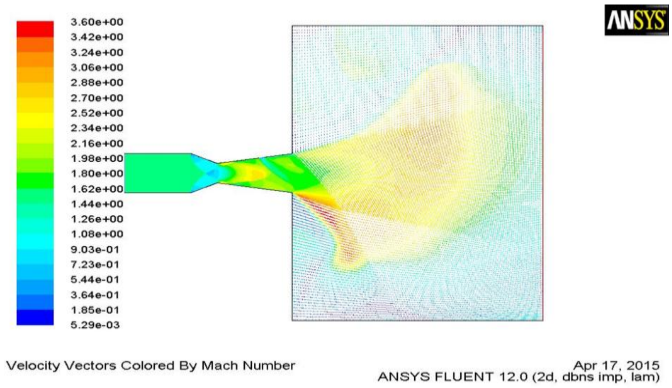
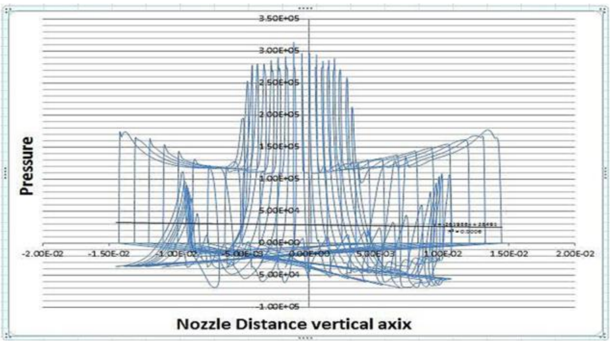


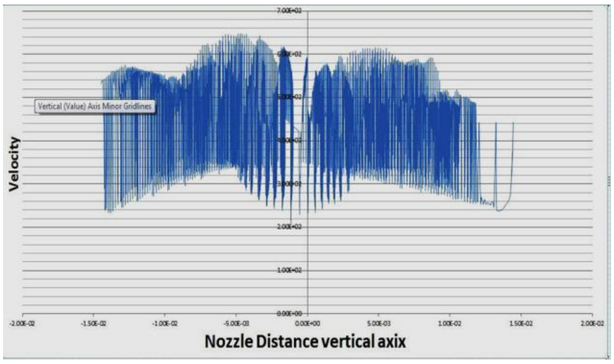
Fig.9. Mach number of C-D nozzle.

In fig 9. Shockwave is created and thus the flow Mach number of the C-D nozzle is converted from subsonic regime to supersonic regime.

Graphical Representation



Graph.1. pressure variation of nozzle.



Graph.2. Velocity variation of nozzle.

From graph, the pressure and velocity reaches a approximate peak level along the nozzle distance along vertical axis indicating a shock creation along the secondary injection in the flow direction.

IV. CONCLUSION

From CFD analysis done on Convergent-Divergent nozzles with varying cross sectional area, the following conclusion has been drawn. At a given operating conditions, maximum thrust Deflection angle was achieved by 18 degrees (i.e) direction of thrust deflected up to 18 degrees in 3D computation for a secondary injection rate of 6%. Increase in the secondary slot width decreased the thrust vector angle without significant efficiency at the 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. Computational results are in good agreement with the experimental and also with the theoretical results published in the literature.

REFERENCES

[1]. Choon Sik Shin, Heuy Dong Kim, Toshiaki Setoguchi and Shigeru Matsuo: Computational Study of Thrust Vectoring Control Using Dual Throat Nozzle, Journal of Thermal Science Vol.19, No.6 (2010) 486−490.

[2]. G.satyanarayana, Ch.varun, S.S.Naiducfd : CFD Analysis of Convergent – Divergent Nozzle, Fascicule 2013, ISSN 2067-3809.

[3]. Subanesh Shyam Rajendran, Aravind Kumar T.R, Nareshkumar K.S : Studies on Thrust Vector Control using Secondary Injection Sonic and Supersonic Jets, (ICMEME'2013) June 17-18, 2013 London (UK).

[4]. Jun-Young Heo, Kwang-Hee Yoo, Yeol Lee, Hong-Gye Sung : Fluidic Thrust Vector Control of Supersonic Jet using Co-flow Injection, AIAA 2009-5174, August 2009, Denver.

[5]. Jerin John, Krishna Ganesh. B, Sanal Kumar V : Numerical Studies on Thrust Vectoring Using Shock Induced Supersonic Secondary Jet, International Journal of Mechanical, Aerospace Engineering, Vol:7 No:8, 2013.

[6]. Andras Sobester, Andy J. Keaney : Multi-objective Optimal Design of a Fluidic Thrust Vectoring Nozzle, AIAA 2006-6916 , September 2006, Portsmouth.

[7]. N. Maarouf, M. Sellam, M. Grignon, and A. Chpoun : Thrust vectoring through fluid injection in an axisymmetrical supersonic nozzle: Theoretical and computational study, ICARE-CNRS, 1C avenue de la recherche scientifique, 45071 Orleans Cedex France.

[8]. Karen A. Deere, Bobby L. Berrier, Jeffrey D. Flamm : Computational Study of a New Dual Throat Fluidic Thrust Vectoring Nozzle Concept, NASA Langley Research Center, Hampton, VA, 23681-000