

# Design of a Parametric Nozzle for Fluidic Thrust Vectoring Experiments

Andrew J. Stagg<sup>1</sup>

The aim of this project is to improve the experimental apparatus for investigation of the shock vector control method of fluidic thrust vectoring in the School of Aeronautical, Civil and Mechanical Engineering at UNSW@ADFA. The primary task in achieving this will be the design of a nozzle in which the geometric parameters can be incrementally altered. A parametric nozzle is essential to experimentally determine the optimum conditions and performance of the shock vector control method. This report will outline the process involved in designing this nozzle.

## Nomenclature

<u>Abbreviations</u>		<u>Symbols</u>	
<i>FTV</i>	: fluidic thrust vectoring	$\eta$	: thrust vector efficiency (°/%-injected flow)
<i>MTV</i>	: mechanical thrust vectoring	$C_{fg,sys}$	: system thrust ratio
<i>SVC</i>	: shock vector control	$\delta$	: thrust vector angle (°)
<i>TS</i>	: throat shifting	$a$	: speed of sound (m/s)
<i>NPR</i>	: nozzle pressure ratio	$M$	: Mach number
<i>MFR</i>	: mass flow ratio (secondary flow)	$p$	: ambient pressure (kPa)
<i>CD</i>	: convergent-divergent	$p_0$	: inlet pressure (kPa)
<i>MATV</i>	: multi-axis thrust-vectoring	$\rho$	: density (kg/m <sup>3</sup> )
<i>LaRC</i>	: NASA Langley Research Center		

## I. Introduction

The emergence of thrust vectoring has enabled significant improvements in combat aircraft performance. It has improved dog-fighting capability by allowing a condition known as supermanoeuvrability, where the conventional flight envelope is extended into the post-stall region. It has extended aircraft range by alleviating trim drag caused by elevator deflection (Mason 2002). It has reduced take-off distances by vectoring thrust downwards on rotation. Thrust vectoring can also be used to reduce the radar cross-section (RCS) of very low-observable aircraft by removing the need for conventional aerodynamic control surfaces (Gal-Or 1989). This concept was proposed for the conceptual X-44 MANTA (Multi-Axis No-Tail Aircraft) (Fig. 1), a long range bomber variant of the F-22 Raptor (Sweetman 2002). These factors combine to give thrust-vectoring aircraft a significant tactical advantage.

In the past, all jet aircraft to apply thrust vectoring have used mechanical thrust vectoring (MTV) techniques. This is done by mechanically deflecting the engine nozzle to direct the flow. Whilst effective, a MTV system is heavy and complex. The MTV nozzle on the F-22A Raptor (Fig. 2) weighs 30% of the total engine weight (Chalmers 1999). To circumvent these problems, fluidic thrust vectoring (FTV) methods have been proposed. FTV is achieved by injecting air into the jet nozzle to create asymmetric flow. It has the advantage of reduced weight, reduced cost and improved reliability. FTV also improves stealth characteristics when compared to MTV by completely removing external moving surfaces (Gamble 2004).



Figure 1. X-44 MANTA (Sweetman 2002)

<sup>1</sup> Undergraduate Aeronautical Engineering Thesis Student (ZACM4050), Australian Defence Force Academy

The methods of FTV that have been researched include shock vector control (SVC), throat shifting (TS), co-flow and counter flow. This project will primarily focus on the SVC method, with the option to modify the experimental apparatus to incorporate TS at a later date. SVC creates an asymmetric vectoring force by turning the flow as it passes through an oblique shock wave. The shockwave is created by injecting a secondary flow at the wall of the divergent section of a convergent-divergent (CD) nozzle. The primary flow experiences a compression ramp at the point of injection of the secondary flow, thus creating a shock wave (Deere 2003).



**Figure 2. Thrust vectoring nozzles on the F-22 Raptor**  
(Jane's 2008)

## II. Background

### A. NASA Langley Research Center

The most extensive research into FTV has been done at the NASA Langley Research Center (LaRC). Research into FTV at LaRC began in 1987 with the SVC method on a 2D CD nozzle similar to the one used at ADFA. This experiment concluded that SVC has the potential to produce large thrust vectoring efficiencies up to  $\eta_v = 4.4^\circ/\%$ -injection but at low thrust ratio of  $C_{f_{g,sys}} = 0.89-0.93$  (Wing 1994).

In 1992, a multi-axis thrust-vectoring (MATV) experiment was conducted combining the SVC method in pitch with the co-flow method in yaw. The co-flow method makes use of the Coanda effect in which the flow adheres to a curved surface due to low pressure caused by high velocity flow near the surface. The experiment concluded that co-flow was not a practical method of thrust vectoring because the flow separated from the Coanda surface at  $NPR > 4$  (Deere 2003).

Throat shifting concept was investigated by the LaRC in 1995. In TS a secondary flow is injected at the throat which turns the flow prior to becoming supersonic. This eliminates the losses associated with shock-waves. The experiment showed TS to be very promising with thrust vector efficiency  $\eta = 1.8^\circ/\%$ -injection and thrust ratio  $C_{f_{g,sys}} = 0.95$  (Deere 2003). An innovative TS method developed at LaRC utilises separation in a recessed cavity. The recessed cavity nozzle concept manipulates flow separation in the recess by injecting air at the throat. Through this method researchers at LaRC were able to achieve large thrust vector angles at high thrust vector efficiency  $\eta_v = 2.15^\circ/\%$  and high thrust ratio  $C_{f_{g,sys}} = 0.96$  (Deere 2003).

Through the investigation done into SVC, TS and co-flow methods of FTV at LaRC, researchers have concluded that TS shows the most potential for application on aircraft. It was found that TS produced the best thrust efficiency but with slightly lower thrust vector angles than SVC. The SVC method was effective but it had poor thrust efficiency because of energy loss through the shock. TS has better efficiency because flow is diverted when it is subsonic. Vector angles achieved through TS are improving with new approaches such as the recessed cavity nozzle.

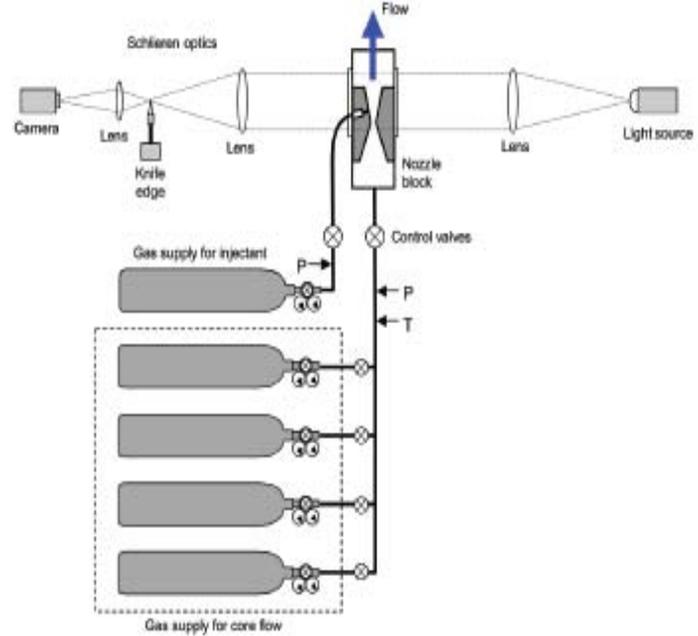
### B. University of New South Wales at the Australian Defence Force Academy

The SACME at UNSW@ADFA began research into FTV in 2005 with the construction of a SVC experimental rig for the undergraduate thesis project by Zhiming Chia (Chia 2005). The SVC method was chosen for the ease of visualisation using a Schlieren optical system. Four 150 MPa compressed air gas bottles provided the primary flow to the nozzle. Another gas bottle was used to supply the secondary flow. The nozzle was a 2D convergent-divergent type with throat area of  $20\text{mm}^2$  and  $NPR = 3.6$ . The secondary flow was injected through a slot in the wall of the divergent section.

The project by Chia successfully recorded Schlieren images of the nozzle with an oblique shock at various angles dependant on NPR and secondary injection pressure. Thrust vector angles were calculated from the Schlieren photos using a MATLAB code developed for the purpose and mass flow was calculated using isentropic flow equation. It was found that the best thrust vectoring angles were achieved when the oblique shock just impinged on the nozzle outlet. Beyond this point the shock reflected off the nozzle wall and turned the flow back towards its original direction (Chia 2005).

In 2007, PhD student Fernando Gesto added load cells to the rig to measure forces directly and obviate the need for the MATLAB code for force calculation. Another series of experiments were conducted with the new apparatus and the results were presented in the paper ‘Performance Studies of Shock Vector Control Fluidic Thrust Vectoring’ by Neely, Gesto and Young (2007). A maximum thrust vector angle of  $\delta = 5^\circ$  was achieved which was significantly less than the thrust vector angles reported in other SVC investigations of up to  $\delta = 15^\circ$ . Numerical simulations conducted by John Young agreed with the maximum degree of thrust vectoring achieved in experiments but over-predicted the mass flow ratio required to achieve optimum thrust vectoring. It was concluded that the thrust vectoring force achieved was not sufficient to justify the use of FTV when compared to the same secondary flow being used for manoeuvring thrusters. The recommendation was made that better performance could be achieved through the use of a nozzle with a lower expansion ratio (Neely 2007).

This project aims to address the problems identified above. Particularly the problems with mass flow measurement and nozzle pressure ratio.



**Figure 3. FTV Experimental apparatus with non-parametric nozzle and Schlieren optical system (Neely 2007).**

### III. Theory

#### A. Flow in a convergent-divergent nozzle

The SVC method of FTV relies on shock-waves to turn the flow. Therefore SVC only works when the flow is supersonic. For supersonic flow in a convergent-divergent nozzle, the flow must become supersonic at the throat. This condition is known as choking and it occurs above a NPR known as the critical NPR. Nozzle Pressure Ratio is defined as  $\frac{p_0}{p}$ . Where,  $p_0$  is the inlet pressure and  $p$  is exit pressure of the nozzle, usually taken as ambient pressure. In the case of the ADFA FTV experiment, the standard atmospheric pressure in Canberra at an altitude of 580 m (BOM 2008) is 94.5 kPa (Anderson 2007). Critical NPR at is calculated from Eq. (1).

$$\frac{p_0}{p_{critical}} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{\gamma}{\gamma - 1}} \quad (1)$$

As the NPR is increased beyond this critical NPR, the velocity at the throat stays at Mach = 1 because it can not go above this value in a convergent duct. Equation (2) governs the flow in a nozzle and explains why a subsonic nozzle is convergent and a supersonic nozzle is divergent.

$$\frac{dA}{A} = (M^2 - 1) \frac{du}{u} \quad (2)$$

At subsonic speeds  $(M^2 - 1)$  is negative so a decrease in area results in an increase in velocity. At supersonic speeds  $(M^2 - 1)$  is positive. Hence, an increase in area results in increased velocity (Anderson 2007).

#### B. Compressible flow

The mechanism for vectoring the flow in the SVC method is an oblique shock. An oblique shock is produced when a supersonic flow experiences an obstruction such as a ramp, causing an instantaneous change in direction, pressure, density, temperature and velocity. The change is instantaneous because ‘information’ can only travel

upstream at the speed of sound. To begin analysing supersonic flow some fundamental values must be calculated such as the speed of sound Eq. (3) and the Mach number Eq. (4).

$$a = \sqrt{\gamma RT} \quad \text{where } \gamma = \frac{c_p}{c_v} \quad (3)$$

Where, R is the gas constant ( $R_{\text{air}} = 287.05 \text{ Nm/kgK}$ ),  $c_p$  and  $c_v$  are specific heat at constant pressure and temperature respectively.

$$M = \frac{v}{a} \quad (4)$$

The equations for an oblique shock are:

$$\tan \theta = \frac{2}{\tan \beta} \frac{M_1^2 \sin^2 \beta - 1}{M_1^2 (\gamma + \cos(2\beta)) + 2} \quad (5)$$

$$\frac{\rho_2}{\rho_1} = \frac{u_1}{u_2} = \frac{(\gamma + 1)M_1^2 \sin^2 \beta}{2 + (\gamma - 1)M_1^2 \sin^2 \beta} \quad (6)$$

$$\frac{p_2}{p_1} = 1 + \frac{2\gamma}{\gamma + 1} (M_1^2 \sin^2 \beta - 1) \quad (7)$$

Where,  $\theta$  is the ramp angle or flow deflection angle and  $\beta$  is the shock angle which can be found from tables at a particular Mach number.

### C. Schlieren visualisation technique

Schlieren imaging is used to visualise the density changes in the nozzle flow. Schlieren works by recording changes in the refractive index ( $n$ ) of a fluid due to changes in its density. For most gases the refractive index is the linear relationship shown in Eq. (4)

$$n = k\rho + 1 \quad (4)$$

Where,  $k$  is the Gladstone-Dale coefficient ( $k = 0.23 \text{ cm}^3/\text{g}$  for air) and  $\rho$  is density.

Schlieren works by projecting a light source through a lens so the light is travelling parallel in the test area. The light is bent by changes in the refractive index of the fluid due to variations in density. This refracted light is then focused by a second lens onto a screen which records the image. But before the light reaches the screen it is passed by a knife edge at the focal point of second lens (Fig. 3). This knife edge blocks any light that was bent downwards. Therefore, areas of different density show up as dark or light areas on the film (Settles 2001).

## IV. Rationale for project

Previous research done using the ADFA FTV experimental apparatus has revealed a number of shortcomings in the design. In the paper by Neely et al (2007), two problems in particular were identified that this project aims to address.

The first problem is the accurate measurement of mass flow. In the past mass flow has been calculated for both the primary and secondary flows using isentropic flow equations. This was done by recording the total pressure in the lines before and after each experiment when there was no flow. Assuming isentropic flow can result in over-predicting the mass flow because losses are not accounted for. This problem will be overcome by installing a mass flow meter on the primary and secondary supply lines, allowing mass flow to be measured directly. An investigation into the most appropriate mass flow meter for the SACME FTV rig was done by Gerard Markham in 2007. He recommended the use of an orifice plate located in a section of pipe of increased diameter, with a AMCA flow conditioner (Markham 2007).

The second problem this project will address is the construction of a parametric nozzle. There were a number of deficiencies with the previous nozzle:

1. The inability to vary the geometric parameters such as NPR and injection location.



Figure 4. Current experimental apparatus set up (Neely 2007).

2. Errors in force measurement caused by the secondary flow hose attachment on the side of the rig (Fig. 4).
3. Separation at the throat caused by sharp throat geometry.

The ability to vary nozzle geometric parameters is necessary to investigate the optimum geometry required to maximise thrust vectoring efficiency. It will also fix the problem with flow separation by allowing different throat geometries. The error in transverse force measurement can be fixed by locating the secondary flow hose attachment parallel to the primary hose. Therefore, the main task of this project is to design a parametric nozzle in which the area ratio and secondary flow injection location can be incrementally altered in a quick and easy manner.

## V. Scope of the Project

The objective of this project is to make improvements to the FTV experimental apparatus at UNSW@ADFA SACME in order to increase the validity and utility of experimental results. The primary task in achieving this objective is to design a parametric nozzle which allows for variation in throat area, exit area and the injection location of the secondary flow. The milestone to gauge the success of this project is to produce drawings of a final nozzle design and make the experimental apparatus operational with the improvements implemented.

To achieve this objective the following tasks must be completed:

1. Conduct a literature review on research done into FTV at ADFA and elsewhere.
2. Investigate 2D and 3D numerical modelling conducted in FLUENT to determine the validity of the 2D assumption.
3. Commission the construction of 4 optical stands to support the Schlieren flow visualisation system.
4. Replace the load cells currently installed in the FTV rig with load cells of the appropriate sensitivity.
5. Select and install a mass flow meter using recommendations from Markham (2007).
6. Conduct a limited set of experiments with the mass flow meter installed.
7. Research requirements for the SACME FTV rig parametric nozzle (Fig. 5).
8. Research different designs of parametric FTV nozzles.
9. Develop a number of conceptual designs for a parametric nozzle.
10. Select the best parametric nozzle design.
11. Develop detailed drawings and workshop procedures for the production of the parametric nozzle.

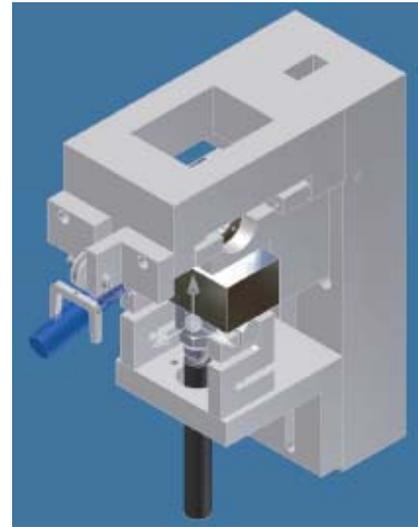


Figure 5. Nozzle block (Neely 2007).

The following tasks are to be completed if time permits:

1. Conduct a preliminary investigation into transient behaviour
2. Build the parametric nozzle and install it in the experimental rig.
3. Conduct preliminary experiments to verify the experimental apparatus is functioning correctly.

## VI. Methods

The principal task in this project is to design a parametric nozzle for the SACME FTV rig. To do this the required nozzle dimensions must firstly be determined. A number of factors will influence the dimensions of the nozzle including 3D effects from the side walls, velocity required to choke the flow and the magnification available in the Schlieren flow visualisation system.

The first step is to evaluate the 3D effects caused by flow interaction with the side walls. This will be done by comparing 2D and 3D CFD models. Slices will be taken of the 3D model in the x-y plane at various distances from the side wall. For these slices, plots will be created of velocity in the y-direction ( $V_y$ ) against y-position at the exit plane and compared with the 2D plot of  $V_y$  vs. y at the exit plane. This will reveal the influence that 3D effects have on thrust vectoring. The same process will be repeated with plots of  $V_x$  vs. y and pressures (p) vs. y to reveal information on the influence 3D effects have on thrust efficiency. Finally a slice will be taken in the y-z plane for  $V_x$  and pressure (p) to graphically represent the influence of the side walls on the flow.

If it is discovered that 3D effects have a significant influence there are two options. The first is to increase the width of the nozzle to mitigate 3D effects. This is not preferable because it will necessitate a smaller nozzle height

for the same area and it creates a nozzle that is less practical for real applications. The second and preferred option is to recommend the use of 3D CFD for future numerical models.

The second step is to choose a nozzle area for the new parametric nozzle. This is influenced by the available mass flow. The option exists to replace the current 3500 kPa regulators with 7000 kPa to increase mass flow. This would be the preferred option if funding is available. Once available pressure is known then calculation of nozzle area to achieve critical pressure ratio for choking can be conducted.

The third step is to determine the design to use for changing the nozzle parameters. To achieve this, research will be conducted on concepts used in other experiments in order to gather a wide range of ideas. Issues such as range of adjustment required and ways of changing and sealing working section will be considered. Conceptual designs of the most promising concepts will be developed, with accompanying lists of pros and cons. From this the best design will be selected.

The final step is to develop a detailed design of the selected nozzle. An engineering drawing will be created in CATIA with workshop instructions to facilitate the construction of the nozzle.

## VII. Work to date

### A. Computational Fluid Dynamics

A three dimensional numerical simulation was conducted by Dr. John Young using the commercial FLUENT CFD program and analysed by the author using the fluids analysis software called TechPlot. The computational domain shown in Fig. 6 includes the convergent nozzle section, divergent nozzle sections, the secondary flow plenum, injection slot and the downstream flow for a limited distance.

The nozzle conditions were calculated by taking a slice of the xy-axis at the centre of the z domain ( $z = 0.04\text{m}$ ). The inlet pressure ( $p_0$ ) was calculated by taking the average pressure of 50 points sampled along the slice at the inlet. The exit pressure ( $p$ ) was assumed to be the international standard atmospheric pressure at sea level.

$$NPR = \frac{p_0}{p} = \frac{2012.485}{94.5} = 21.3$$

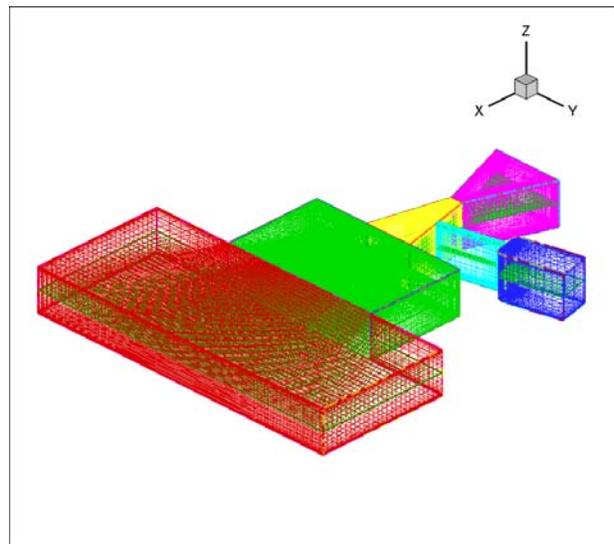


Figure 6. Computational domain of 3D simulation.

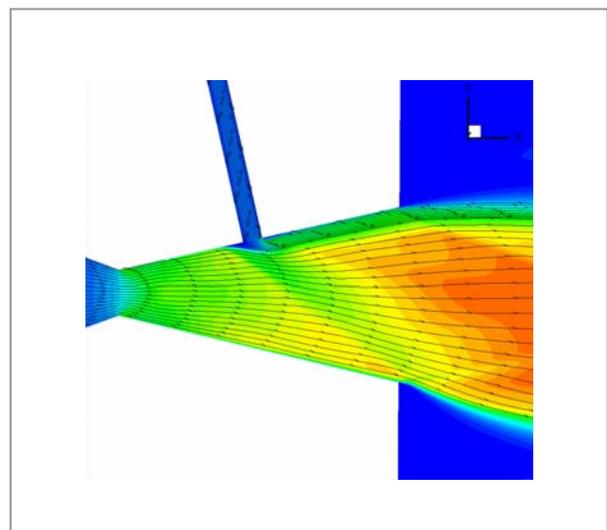
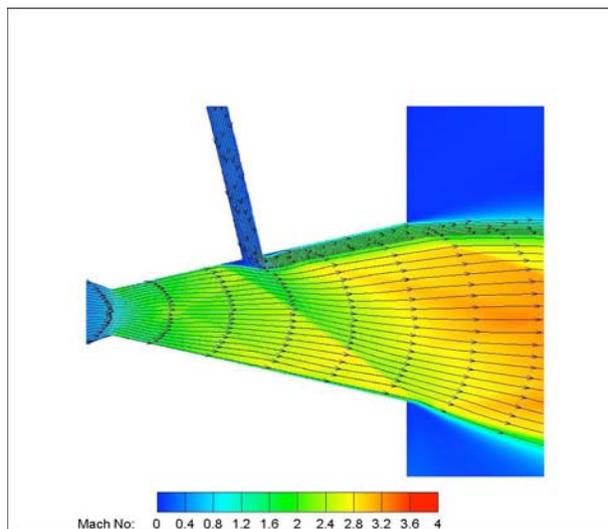


Figure 7. Mach contours and streamlines of (a) 2D model at NPR=20, MFR=0.07 and (b) 3D model at NPR=20

The 3D simulation conducted at  $NPR \approx 20$  was compared against the 2D simulation done previously at  $NPR = 20$  and  $MFR = 0.07$  (Neely 2007). The plot of Mach contours for the 2D Fig. 7(a) and 3D Fig. 7(b) simulation are shown below. It can be seen that the flow structures are similar except that the Mach numbers calculated in the 3D model are much higher. The maximum Mach number recorded in the 3D model was  $M=59.5$ , which is completely unrealistic. The reason for the unrealistic values of Mach number was a calculation error in TechPlot. Temperature was calculated at approximately 120 K which caused the speed of sound calculation to be wrong. This problem was not encountered when the properties were checked in Fluent, so it was concluded that the fault was in TechPlot. This conclusion is supported by realistic values of velocity magnitude in the region of 600 m/s in the domain.

In the 2D CFD investigation, plots were made of the axial (x) velocity (Fig.8), normal (y) velocity (Fig. 9) and pressure (Fig. 10) across the exit plane as a measure of nozzle performance (Neely 2007). Equivalent plots have been created for the 3D model and compared against data extracted from the 2D model plots.

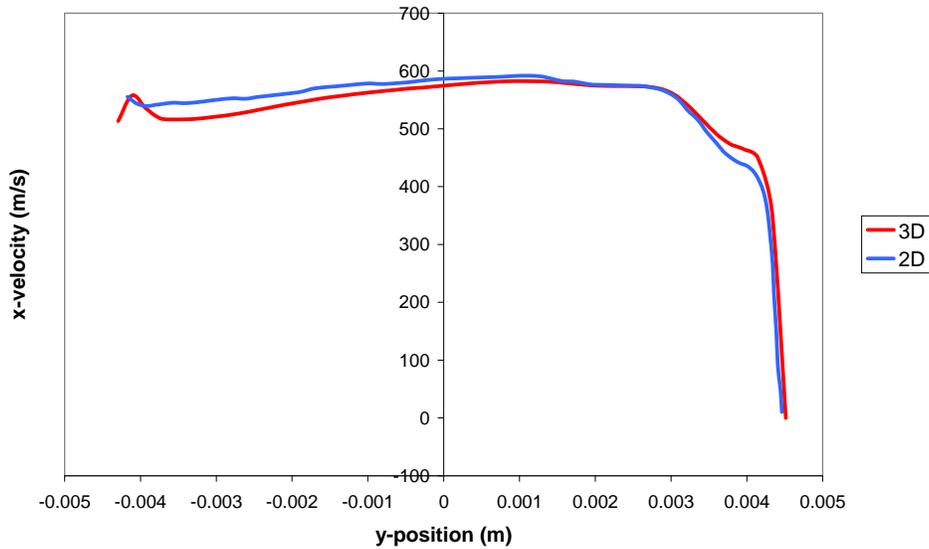


Figure 8. Plot of x-velocity across the exit plane.

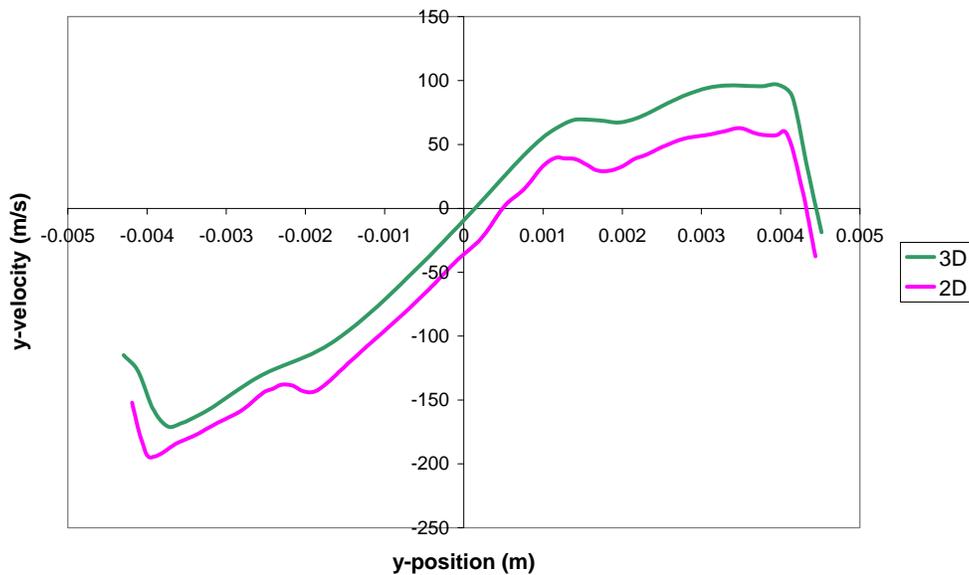
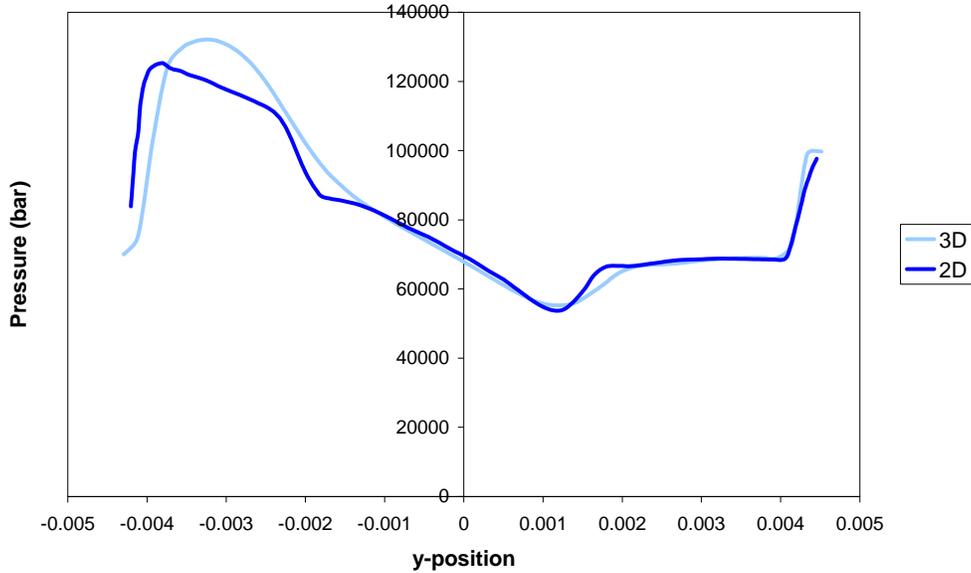


Figure 9. Plot of y-velocity across the exit plane.

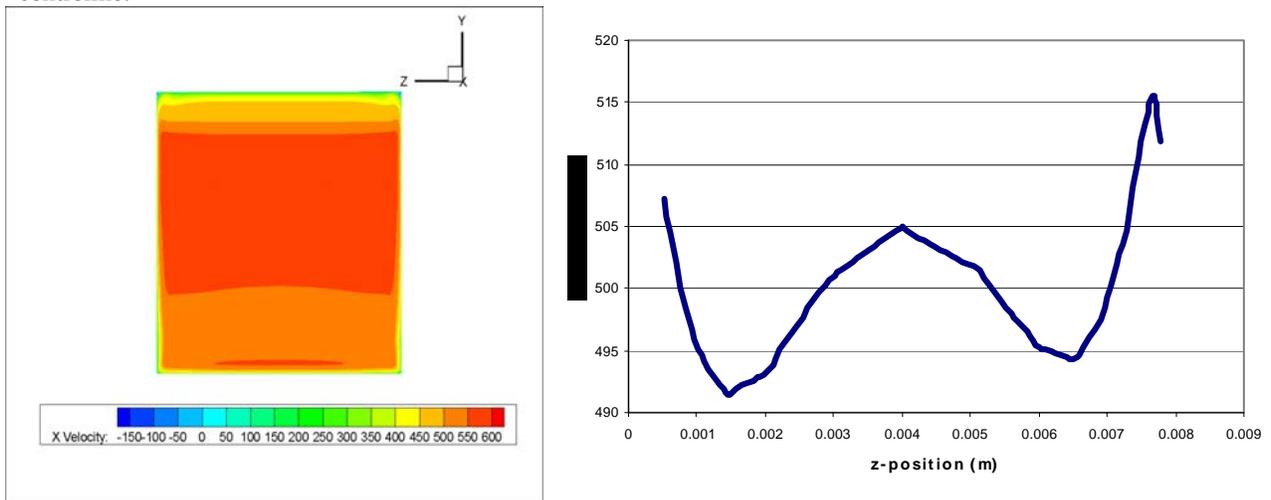


**Figure 10. Plot of pressure across the exit plane.**

Momentum thrust was calculated by multiplying local density and local velocity squared at discrete points then integrating across the exit plane. This was done for the axial and normal directions to give momentum thrust and momentum vectoring force respectively. The pressure thrust was also calculated by integrating the product of pressure and area across the exit plane (Neely 2007).

It can be seen that the 2D model approximates the 3D model reasonably well except for in the case of normal momentum thrust. In this case the 2D curve is below the 3D curve, resulting in a more negative total area between the curve and the axis. This means the 2D model will give more normal momentum thrust than the 3D model.

Figure 11(a) shows a slice taken parallel with the nozzle exit plane with contours of axial velocity. This view is very informative when analysing the 3D effects of the side walls. A thin boundary layer can be seen on each side wall which is much thinner than the boundary layer at the top. The flow is close to symmetrical about the vertical centreline.



**Figure 11. (a) Slice of exit plane showing x-velocity contours. (b) Plot of x-velocity across nozzle exit plane.**

Figure 11(b) is plot of x-velocity across the horizontal centreline of exit plane. The flow is clearly not constant across the domain. Thrust is the integral across the area so any thrust calculation based on the velocity at the centreline of 505 m/s would be inaccurate. However, the magnitude in variation is relatively small when compared to the variation in x-velocity along the y-axis. The asymmetry is likely due simulation not having done enough iterations to fully converge.

In conclusion, the 2D numerical simulation is sufficient for most work such as determining trends and optimising nozzle conditions, but the 3D numerical simulation would be required for calculating quotable performance values.

## B. Parametric Nozzle Conceptual Drawing

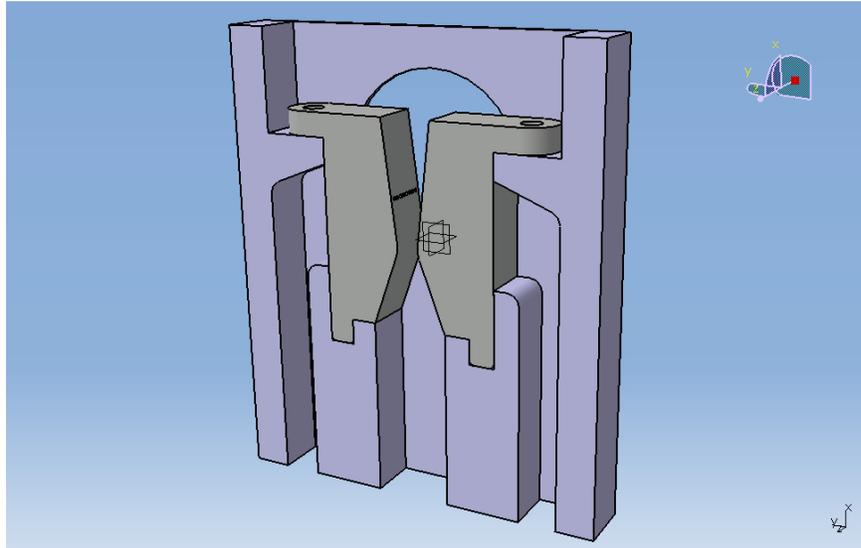


Figure 12. Conceptual drawing of the parametric nozzle.

A conceptual drawing of the parametric nozzle was made in CATIA (Fig. 12). It incorporates an interchangeable nozzle area to facilitate the variation of geometric parameters. The nozzle is secured by slots and two bolts, which are intended to ensure sealing. The secondary flow enters at the bottom to prevent errors in the transverse load readings. Secondary flow ports were incorporated in both sides to facilitate future expansion to include fluidic throat area control experiments.

## VIII. Summary

This report has outlined the planned approach to addressing the limitations of the SACME FTV experimental apparatus and conducted a preliminary investigation into the experimental requirements. The analysis done on the validity of the CFD used to model flow in the nozzle concluded that 2D CDF is appropriate in most circumstances. Finally a conceptual design was produced as a starting point for the design.

Fluidic thrust vectoring is a cutting-edge area of research which has the potential to give a significant tactical advantage to combat aircraft. The FTV experimental apparatus at UNSW@ADFA gives members of SACME the opportunity to be part of this research. But to contribute effectively to the research the ADFA FTV rig needs to be upgraded to address two key limitations. These being the ability to measure mass flow and the ability to change the nozzle geometric parameters. This project will address both, therefore improving the validity and utility of experimental results.

## References

Anderson, J. D. (2007). Fundamentals of Aerodynamics, McGraw-Hill.

BOM. (2008). "Climate of Canberra area." Retrieved 01 May 08, from <http://www.bom.gov.au/weather/nsw/canberra/climate.shtml>.

Chalmers, P. (1999). "Return Flight." ASME.

Chia, Z. (2005). An experimental investigation of fluidic thrust vectoring via the shock vector control method. School of ACME. Canberra, UNSW at ADFA. **B Eng AERO**: 115.

- Deere, K. A. (2003). Summary of Fluidic Thrust Vectoring Research Conducted at NASA Langley Research Center. 21st AIAA Applied Aerodynamics Conference. Orlando, Florida, NASA Langley Research Centre. **AIAA-2003-3800**: 18.
- Gal-Or, B. (1989). *Vectored Propulsion, Supermaneuverability and Robot Aircraft*, Springer-Verlag.
- Gamble, E., Terrell, D., DeFrancesco, R. (2004). "Nozzle Selection and Design Criteria." AIAA 2004(3923): 11.
- Jane's (2008). Lockheed Martin (645) F-22 Raptor. Jane's All The Worlds Aircraft, Jane's.
- Markham, G. (2007). Selection of Mass Flow Measuring Device for the UNSW@ADFA Fluidic Thrust Vectoring Experimental Equipment. CDF Project. Canberra, UNSW at ADFA: 58.
- Mason, M. S., Crowther, W.J. (2002). Fluidic Thrust Vectoring of Low Observable Aircraft. CEAS Aerospace Aerodynamics Research Conference. Cambridge, UK, University of Manchester, School of Engineering: 7.
- Neely, A. J., Young, J and Gesto, F.N (2007). Performance Studies of Shock Vector Control Fluidic Thrust Vectoring. AIAA, UNSW@ADFA: 14.
- Settles, G. S. (2001). Schlieren and Shadowgraph Techniques: Visualizing Phenomena in Transparent Media, Springer.
- Sweetman, B. (2002). Raptor Could Hatch a Delta Bomber. Jane's International Defence Review, Jane's.
- Wing, D. J. (1994). Static Investigation of Two Fluidic Thrust-Vectoring Concepts on a Two-Dimensional Convergent-Divergent Nozzle. NASA Technical Memorandum. Hampton, VA, NASA Langley Research Center.

### Appendices

- A. Client brief
- B. Project management documentation
- C. CFD flow condition spreadsheets