Throat Shifting Method of Fluidic Thrust Modulation (FTM) & Fluidic Thrust-Vectoring (FTV)

J. Y. Lim

University of New South Wales at the Australian Defence Force Academy

Thrust vectoring and control is one of the most significant design enhancements of high-performance aircraft manoeuvrability adopted by aircraft designers today. Whereas previous mechanical methods for controlling and vectoring the thrust have proven to be complex and costly, fluidic thrust vectoring has the potential to be a cheaper and simpler alternative. In recent years, fluidic thrust vectoring and modulation have generally been developed into four different methods. This report reviews the four different methods of thrust vectoring, and develops an understanding for the significant factors that affect one of the less well-known methods of fluidic thrust vectoring and control – the throat shifting method. The thesis adopts a design of experiments approach to calculate the significance of the factors, and obtains a detailed understanding of how the factors interact with each other to produce the best throat shifting nozzle performance in terms of thrust vectoring and thrust modulation with the lowest secondary mass input required.

Contents
Nomenclature ........................................................................................................................................... 2
I. Introduction .................................................................................................................................................. 3
II. Background .................................................................................................................................................. 3
   A. Background of Fluidic Injection Methods ......................................................................................... 4
      1. Co-flow Method .................................................................................................................................. 4
      2. Counter-flow Method ......................................................................................................................... 5
      3. Shock Vector Control (SVC) Method ................................................................................................. 5
      4. Throat Shifting (TS) Method ........................................................................................................... 6
   B. Comparison of Fluidic Injection Methods ......................................................................................... 7
III. Aim of Thesis ........................................................................................................................................... 8
IV. Outline of Thesis ....................................................................................................................................... 8
V. Thesis Methodology ..................................................................................................................................... 8
   A. Planning Phase ....................................................................................................................................... 9
      1. Selection of quantifying variable ....................................................................................................... 9
      2. Selection of factors ............................................................................................................................. 10
      3. Setting levels of factors ...................................................................................................................... 10
   B. Designing phase ..................................................................................................................................... 11
   C. Conducting Phase ................................................................................................................................. 11
      1. CATIA/GAMBIT mesh generation .................................................................................................. 11
      2. FLUENT case setup ......................................................................................................................... 13
      3. CFD Verification ................................................................................................................................ 14
      4. CFD Validation .................................................................................................................................... 15
   D. Analysis Phase ....................................................................................................................................... 16
      1. Main effect plots ............................................................................................................................... 16
      2. Interaction plots ............................................................................................................................... 16
      3. Pareto plots ....................................................................................................................................... 17
VI. Initial Computational Testing .................................................................................................................. 17
   A. Mesh Grid dependence study ........................................................................................................... 17

1 OFFCDT, School of Aerospace, Civil & Mechanical Engineering. ZACM 4049 Aeronautical Engineering: Project, Thesis and Practical Experience A

Final Thesis Report 2009, UNSW@ADFA


Nomenclature

\( A^* \) = nozzle throat area
\( AR \) = nozzle aspect ratio (nozzle depth/nozzle width)
\( A_{exit} \) = nozzle exit area [mm\(^2\)]
\( A_{exit}/A^* \) = nozzle area ratio
\( A_{inj} \) = injector slot area [mm\(^2\)]
\( A_{ratio} \) = area ratio, \( A_{inj}/A^* \)
\( C_{fg} \) = gross thrust coefficient of nozzle primary outflow
\( \delta_{ps} \) = primary nozzle outflow deflection angle in yaw
\( \varepsilon \) = effect of varying a factor on effectiveness
\( \dot{E}_T \) = dissipation rate of turbulent kinetic energy [m\(^2\)/s\(^3\)]
\( \eta_{FTV} \) = measure of effectiveness for TS FTV
\( \eta_{FTM} \) = measure of effectiveness for TS FTM
\( \gamma \) = ratio of specific heats
\( h_{ut} \) = upstream throat height of dual-throat nozzle [m]
\( h_{dt} \) = downstream throat height of dual-throat nozzle [m]
\( k \) = turbulent kinetic energy [m\(^2\)/s\(^2\)]
\( l \) = cavity length of dual throat nozzle [m]
\( M_e \) = Mach number at the nozzle exit
\( m_p \) = primary mass-flow rate through nozzle [kg/s]
\( m_s \) = injected mass-flow rate from injector into nozzle [kg/s]
\( p_{air} \) = air pressure [Pa]
\( p_{atm} \) = atmospheric (environmental) pressure [Pa]
\( p_{exit} \) = mass-weighted average pressure at nozzle exit [Pa]
\( p_{pri} \) = primary inlet pressure [Pa]
\( Pr \) = injector-primary mass flow ratio, \( p_{inj}/p_{pri} \)
\( p_{exit} \) = air pressure at exit [Pa]
\( p_{ref} \) = reference air pressure [Pa]
\( p_{inj} \) = injector inlet pressure [Pa]

Final Thesis Report 2009, UNSW@ADFA
\( p_t \) = total pressure at nozzle inlet [Pa]
\( r_G \) = grid refinement ratio
\( T_0 \) = net thrust of nozzle without injection [N]
\( T_{nj} \) = net thrust of nozzle with injection [N]
\( T_{net} \) = net thrust of nozzle [N]
\( \theta_{nj} \) = angle of throat injection [degrees]
\( \theta_1 \) = nozzle diverging angle downstream of throat [degrees]
\( \theta_2 \) = nozzle converging angle downstream of throat [degrees]
\( v_{aircraft} \) = velocity of aircraft [m/s]
\( v_x \) = mass-weighted average of x-component of velocity at nozzle exit [m/s]
\( v_y \) = mass-weighted average of y-component of velocity at nozzle exit [m/s]
\( w_e \) = nozzle exit width [mm]
\( w_t \) = nozzle throat width [mm]

I. Introduction

The demand for aerospace power requires technological advancements in the design of high-performance aircraft. For the aircraft to operate at high levels of performance, the nozzle geometry needs to be continuously optimised for changes in the Mach number and engine pressure ratio (Deere and Hunter 1999). In the optimisation of nozzle geometries to achieve high-performance, mechanical actuators have been fitted on the nozzles to change nozzle throat and exit areas (Federspiel et al. 1995). By manipulating the nozzle throat and exit areas, the mass-flow rate through the nozzle can be varied, which potentially varies the thrust of the aircraft – this process is described as thrust modulation (TM). In addition, the skewing or shifting of nozzle throat and exit areas causes the mass flow and the thrust at the nozzle exit to be vectored, providing an additional side force which increases the manoeuvrability of the aircraft – this process is termed thrust vectoring (TV).

Research into TV has successfully identified and demonstrated significant potential benefits to high-performance aircraft that contribute to aerospace power (Kowal 2002). In comparison, less research has been conducted on the process of TM; however, the combination of TV and TM in the nozzle potentially allows for a significant increase in the manoeuvrability of high-performance aircraft, and is worth investigation.

The problem of using mechanical actuators to achieve TV or TM in nozzles, however, is that the designs are highly complex and have heavy weight requirements. Recent research into the use of a fluidic (gaseous) injection system in the nozzle to replace the mechanical actuators has proposed solutions to overcoming these pitfalls of the mechanical system, without major compromises in aircraft-performance enhancements (Kowal 2002). Compared to the mechanical system, the use of a fluidic injection system has the advantage of reduced complexity, decreased weight, reduced operating costs and increased reliability. In addition, fluidic injection systems have improved stealth characteristics due to the removal of external moving surfaces. The fluidic injection system is therefore potentially a more suitable candidate for nozzle designs on high-performance aircraft (Gamble et al. 2004).

Despite extensive research being performed on the fluidic injection system, there are very few papers that attempt to explain the fundamental relationships between the different parameters that are considered in designing a fluid injector in the nozzle. These parameters, for a given nozzle geometry, are that of the angle of fluidic injection, the nozzle pressure ratio, the ratio of primary flow pressures to injected flow pressures and the injection slot size. Through the use of 2D computational fluid dynamics (CFD), this paper investigates the relationships between these parameters, and aims to provide a basic understanding in using a fluidic injection system to achieve TV and TM.

II. Background

Most of the research into the fluidic injection system has been in the use of fluidic injection systems for TV, a method referred to as fluidic thrust vectoring (FTV). There are few papers that directly address the possibilities of using the fluidic injection system for thrust modulation, a process described as fluidic thrust modulation (FTM). The following sections review the literature available involving different methods for FTV or FTM.
A. Background of Fluidic Injection Methods

Thrust vectoring is a control method employed by high-performance aircraft that manipulates the nozzle flow to increase aircraft manoeuvrability and performance. Recent research into nozzle enhancements proposed the use of a secondary injection of fluid into the nozzle to induce a change in the nozzle outflow axis, therefore achieving FTV. In general, the research and development into fluidic injection into the nozzle can be divided into 4 different methods, namely the co-flow method, counter-flow method, shock vector control method, and throat shifting method.

1. Co-flow Method

The co-flow method is mostly used for FTV, and relies on a phenomenon known as the Coanda effect, which is the tendency of a fluid and gaseous jet to adhere to a convex curvature of a solid boundary (Banazadeh et al. 2008). The conceptual setup of a co-flow TV nozzle is shown in Figure 1 below.

The secondary bleed air from the engine is injected along the side of the primary jet nozzle outflow. The pressure and velocity difference between the two jets forms a convex surface relative to the primary jet, triggering a vectoring of the primary jet outflow. There is also added vectoring as the secondary jet attaches to the diverging curvature at the exit of the nozzle; an area described as the nozzle collar (Sobester and Keane 2006).

Co-flow FTV has typically three zones of TV performance, which can be observed in Figure 2 (Banazadeh et al. 2008). The plot shows the effect of varying secondary flow on the TV angle for a range of revolutions per minute (RPM) of the engine that Banazadeh investigated. It is noted that although the results are dependent on the geometry or size of the engine under investigation, the analysis serves as an example of how there are typically zones of unreliability of the co-flow method of FTV.

The first zone across the range analysed is described as the ‘Dead Zone’, which exhibits insignificant FTV performance, and occurs at low secondary flow rates, with constant low thrust vectoring angles of about 5 to 10 degrees. This result is attributed to the separation of primary jet thrust from the original flow, which reduces the effectiveness of the co-flow injection as a Coanda surface.

The second zone is the ‘Active Zone’, which is a zone where TV angles increase with secondary flow rate. In this zone TV angles of 10 to 20 degrees for 20 to 40 grams/sec of secondary flow, depending on engine RPM. This zone reflects the region where the Coanda effect is optimal.

The last zone, the ‘Saturation Zone’, occurs where further increases in secondary flow fails to increase TV angles. In some cases further injection causes a decrease TV angles. This is a likely result of physical limitations where the vectoring angle cannot exceed the geometry of the collar surrounding the nozzle outflow.

Research into co-flow FTV has highlighted several critical design parameters for co-flow FTV, ranging from nozzle geometries to jet outflow velocity profiles (Sobester and Keane 2006), which potentially reduces pressure losses of the co-flow FTV method expected from the ducting of the secondary flow.

The problem with the co-flow FTV is that of hysteresis effects and pressure losses that occur when the primary jet outflow attaches to the nozzle collar. The flow attachment to the collar causes the system to be irreversible, and requires the designer to design additional mechanisms to the structure in order to restore the system to its original state of nozzle flow. This additional weight and complexity is highly undesirable in
aircraft. The method also exhibits instability in the ‘Dead Zone’ and the ‘Saturation Zone’; therefore this design is not favourable in the purposes of FTV and FTM combinations.

2. Counter-flow Method

The counter-flow method is also based on the Coanda effect as previously described in the co-flow method. Similar to the co-flow method, majority of the work on the counter-flow method is based on its FTV capability. In the case of counter-flow FTV, the convex surface required to induce the Coanda effect is created from a reversal of secondary flow direction relative to the primary nozzle outflow. A sketch of a counter-flow FTV setup is shown in Figure 3.

In order to reverse the secondary flow relative to the primary jet, a vacuum is generated in a slot adjacent to the primary jet (Flamm 1998). Two curved surfaces, also known as collars, are introduced around the jets to create the envelope for TV operation. Suction is then applied to the plenum chamber in the required nozzle side in order to induce a pressure gradient, which vectors the primary flow as a result of the Coanda effect. (Van der Veer and Strykowski 1997).

Theoretically, counter-flow FTV requires less secondary flow than co-flow FTV to vector the primary jet outflow as the opposition of the secondary flow causes a more significant pressure gradient between the flows which aids in directing the primary flow. This added benefit of the counter-flow method allows more air to be directed through the engine (instead of bleeding air for secondary flow) and increasing thrust. Counter-flow experiments also have the added effect of cooling of primary jet outflow due to the added interaction of cool ambient air flow (Washington and Alvi 1996).

Counter-flow experiments, however, are also subjected to problems of instability previously highlighted in the co-flow method due to the passive nature of the Coanda effect. There are similar problems in that the primary jet outflow attaches to the suction collars at certain conditions (Deere 2003).

In the example shown in Figure 4, stable operation is not possible when pressure difference between the jets lie in the region 50 to 90 Torr. This is typical for FTV nozzles using the counter-flow method.

Also, integrating the counter-flow method into existing aircraft nozzles is a more complex process than the other fluidic injection systems, because the counter-flow method requires an additional source for generating suction, which potentially requires the inclusion of an additional system around the nozzle (Deere 2003).

3. Shock Vector Control (SVC) Method

The SVC FTV method uses the generation of oblique shock waves in the nozzle divergent section to turn the flow in the nozzle (Wing 1994). An example of a planar 2D flow structure in a nozzle applying the SVC method is shown in Figure 5.

The injected flow behaves like a ramp obstacle in the primary flow, causing compression of the oncoming flow and creating an oblique shock wave. The primary flow interaction with the oblique shock wave generated directs the jet outflow away from the longitudinal axis of the aircraft, therefore achieving TV.

As the secondary mass flow injected increases, the

![Figure 3. Schematic of FTV nozzle with counter-flow in upper shear layer of jet (Flamm 1998)]

![Figure 4. Counter-flow TV angle against pressure difference across primary and secondary jet. (Van der Veer and Strykowski 1997)]

![Figure 5. SVC FTV setup (Deere 2003)]
angle of the oblique shock is turned further away from the injector nozzle flap. Optimal TV is achieved when the oblique shock just impinges on the opposite nozzle flap (Neely et al. 2007). The increase in pressure of the primary flow across the oblique shock results in significant decreases in jet pressure thrust (Ko and Yoon 2002).

Further increases in the injected flow beyond optimal levels result in the oblique shock impinging further forward on the opposite nozzle flap, causing the generation of a reflected shock. The reflected shock has an opposite effect to the oblique shock, turning the primary flow back along the axial direction.

An experiment conducted by Ko and Yoon on 3D SVC FTV (Ko and Yoon 2002), however, has shown that SVC FTM is the net result of two conflicting phenomenon: the reduction of pressure thrust after the oblique shock induced and the increase in momentum thrust because of added mass flow from secondary injection. Figure 6 shows that the amount of augmented axial thrust increases with increasing ratio of secondary to primary mass flow injected, with the 3 different lines reflecting different nozzle geometries. Ko and Yoon attributed the increase in thrust to the addition of the secondary mass flow into the axial mass flow, which increases the total mass flow at the nozzle exit. Ko and Yoon concluded from their experiments that when the oblique shock impinges on the opposite nozzle surface, the reflected shock generated turns the flow towards the axial direction, which increases the axial momentum thrust and decreases the amount of FTV achieved. Review of the research results from Neely (Neely et al. 2007) also suggest that the SVC method is capable of achieving highest FTV angles when the oblique shock just impinges upon the opposite nozzle flap. For FTV applications at sub-optimal conditions, where the oblique shock does not impinge on the opposite nozzle face, the thrust is expected to decrease as a direct result of the loss of total pressure and nozzle flow velocity across the oblique shock (Deere 2003).

At the time of this paper, PLTOFF Brendan Blake, a fellow undergraduate, is developing a thesis paper on the FTM effects of SVC. The initial meshes and CFD results have been shared with thesis supervisors, Dr Andrew Neely and Dr John Young, who will then extend these results into a paper for future PhD students continuing the experiments in UNSW@ADFA.

4. Throat Shifting (TS) Method

Throat shifting uses the shifting of the nozzle throat geometry to manipulate flow separation and cause thrust vectoring of the primary jet thrust (Deere et al. 2003). An example of throat shifting nozzle setup is as shown in Figure 7. For the TS method, the injectors are located at the throat of the nozzle, where the cross sectional area of the nozzle is at its minimum. In order to achieve FTV, the injection needs to be asymmetric; as such the injection is modelled through the positioning of injectors only at the bottom wall of the nozzle as shown in Figure 7.

The injection of secondary mass flow at the throat shifts the sonic plane sub-sonically, as the injected flow interacts with the primary flow by primarily reducing its cross-sectional flow area (through jet penetration) and also mixing with the primary flow through the development of shear layers between flows. The combination of these effects results in the reduction of the nozzle throat from a geometrical minimum to a new minimum caused by the reduction of primary-flow cross sectional area at the throat due to injected-jet penetration – an aerodynamic minimum as described by Deere (2003).

Sub-sonic shifting of the sonic plane implies that TS FTV does not induce a loss of total pressure in the nozzle that would have been present in the SVC method of FTV. Comparative studies between the SVC and TS
method, however, have shown that TS FTV method are capable of achieving smaller TV angles compared to the SVC FTV method (Deere, 2003).

Comparatively, less research has been performed involving the FTM possibilities of TS. The TS FTM method relies on the use of injectors around the throat of the nozzle injecting symmetrically to decrease the throat area from a geometric minimum to an aerodynamic minimum. Figure 8 shows two examples of TS FTM experiments with the secondary injector injecting at 60 degrees and 30 degrees (Federspiel et al. 1995). Federspiel performed a study of the effects of the parameters of secondary flow rates, injection slot sizes, the injection angle and the ratio of primary nozzle stagnation pressure to atmospheric pressure (defined as the nozzle pressure ratio, NPR). Federspiel concluded that a reduction of 70% of the primary flow rate is possible from the TS method of FTM. The loss of thrust, however, were discovered to be small at the NPR where the experiments were conducted, which were intended as a reflection of the NPR at which aircraft nozzles would operate at in high-performance flight.

Most of the other research into the TS method involved FTV experimental runs conducted with varying nozzle geometries in the NASA Langley Research facility (Deere 2003). The research, however, cites results from experimental runs with varied nozzle geometry without identifying the significance of the variations in injector geometries and operating conditions used in nozzle design or the possible interaction between these parameters.

Therefore, there is a case for the investigation of the selection of the above-mentioned parameters and the development of a fundamental understanding of the significant relationships between different parameters of the nozzle using the TS method. This is an important aspect of research that enables developers to understand how each nozzle parameter will influence the nozzle’s TS FTV capability, and how operating conditions may affect nozzle performance. A parametric study of the nozzle needs to be conducted to pave the next step into TS FTV and FTM applications.

B. Comparison of Fluidic Injection Methods

A summary of the fluidic injection methods used, results obtained, and reference values used in the papers the author reviewed can be found in Appendix A. This background study and summary table will be used in the comparison of the nozzle performance at the end of the design process performed by the author.

Although co-flow and counter-flow FTV methods are commonly mentioned through summary articles, the methods are often quoted to be subjectable to instability in certain ranges of FTV, and subjected to hysteresis effects and associated losses as previously discussed. In comparison, SVC and TS have been accepted to be the two methods with highest potential for aircraft application and integration as these methods provide better efficiency and effectiveness in thrust vectoring, and are considered the more reliable methods of FTV.

Throat shifting FTV has the benefits of achieving multi-axis TV control with generally higher thrust efficiency than that of SVC FTV (Yagle et al. 2001). Figure 9 compares different FTV methods at NPR of 5, where LM Aero CRN is the TS FTV facility used by Yagle (2001). The TS FTV setup (LM Aero CRN) is more efficient at conserving thrust in the range showed plotted on the
diagram - Cfg values are 3% higher than the SVC FTV method that achieves largest deflection angles ($\delta_y$), where the ratio of nozzle exit area to nozzle inlet area ($A_9/A_8$) = 1.944 (Yagle et al. 2001). It is worth noting, however, that the TS FTV method is not capable of achieving the same level of thrust deflection angles as the SVC methods, with the LM Aero CRN setup reaching a highest $\delta_y$ value slightly more than 8 where SVC methods with $A_9/A_8 = 1.944$ and 2.405 achieved $\delta_y$ values of approximately 11. This incapability of TS FTV in achieving high FTV angles is consistent with the results observed by Deere (2003). In the results of Yagle’s experiments, in particular, there is an additional observation in that for TS FTV, there is decreasing thrust efficiency $C_{fg}$ as the value of $\delta_y$ increases. In comparison, SVC FTV methods with $A_9/A_8 = 1.944$ and 2.405 do not appear to exit this behaviour, in fact the setup with $A_9/A_8 = 2.405$ has increasing $C_{fg}$ as achieved $\delta_y$ increases. This experimental result suggests that if TS FTV would achieve lower $C_{fg}$ at higher values of $\delta_y$ ($\delta_y$ more than 10) – this trend needs to be investigated in future comparative studies between high-performance TS FTV and SVC FTV setups.

In the aerospace power context it is important that high-performance aircraft have sufficient thrust to execute and switch between manoeuvres when required. Although SVC FTV currently exhibits the potential for allowing the aircraft nozzles to provide vectoring for manoeuvres at off-design conditions, TS FTV has the potential of achieving the same manoeuvres without the inherent loss of thrust and outflow velocity through the manoeuvre. It is therefore worth investigating methods for enhancing TS FTV, especially aiming to achieve higher TV angles without sacrifices to the thrust of the aircraft.

In contrast, in rocket nozzle applications, it is important that thrust can be reduced, or modulated, to achieve better performance and control of the aircraft. Compared to the SVC method, there is an understanding that the TS method is capable of reducing thrust more consistently in TM applications. There is, however, a considerable gap in the understanding of the fundamental relationship between different nozzle design parameters (listed previously), and as such a parametric study also needs to be conducted into the TS method before making further enhancements.

### III. Aim of Thesis

The aim of this thesis is to identify the nozzle-injector geometry and secondary-injection operating requirements that achieves the highest thrust modulation or highest thrust vectoring for the lowest amount of fluidic injection using the throat shifting method in a convergent-divergent nozzle.

### IV. Outline of Thesis

This thesis uses the design of experiments approach described by Anthony (2003) to conduct 2D planar parametric studies of the TS FTV and FTM nozzle geometry and secondary injector locations. The CFD method used from numerical studies will be validated against experimental results obtained from the UNSW@ADFA FTV nozzle experimental setup. A copy of the client brief and the detailed thesis task lists are located in Appendix B and Appendix C respectively. It should be noted, however, that since the client brief developed the thesis has been modified, and as such two of the deliverables have been replaced by the parametric study of the TS FTM method. The milestone chart for the thesis was monitored and can be found in Appendix D. Upon the decision to conduct an extended 2D parametric study, a Gantt chart was reproduced to manage the time available to the author in the time available in Session 2, 2009. This modified Gantt chart is put together with the original Gantt chart developed at the time of the initial thesis report in Appendix D.

The 2D parametric study is conducted by varying the factors of injector-slot area to nozzle throat area ratio, secondary-injection angle, primary flow to secondary flow pressure ratio and nozzle pressure ratio. The basic nozzle geometry is a conventional convergent-divergent nozzle based on the Deere (2003) nozzle that achieved the highest FTV efficiency. Upon completion of the design of experiments approach, the interactions between the above-mentioned factors and also their effects on FTV and FTM operations will be discussed, and an optimum nozzle-injector design for FTV and FTM operations can then be identified.

### V. Thesis Methodology

For the purposes of this thesis, the author employs an approach, described in the sections below, that follows a full factorial experimental design as proposed by Jiju Anthony in his book ‘Design of Experiments’ (Anthony 2003). The full factorial experimental design approach helps the designer to understand the significance of factors and their mutual interactions. Both TS FTM and TS FTV will be studied with the full factorial experimental design approach with the factors as previously described in the ‘Thesis Outline’ section.
This thesis employs FLUENT, a CFD modelling software, to perform computational simulations of a convergent-divergent nozzle with secondary mass flow injected at the nozzle throat in a planar 2D setup. The mesh-grid used for the simulations are drawn using CATIA Version 5, a computational drawing software, and then meshed using GAMBIT, a grid meshing software, before importing into FLUENT.

A. Planning Phase

The experimental approach in this section follows the guide proposed by Anthony (2003). The experiment in this thesis is defined as the 2D simulation of the flow within a nozzle that has both primary flow entering the nozzle from the engine and also secondary flow injected at the throat through secondary flow injectors. The experiment conducted with the flow entering the nozzle as an ideal gas at room temperature to facilitate computations and validation using the UNSW@ADFA FTV setup, which is described in the ‘Validation’ section of this thesis.

In the planning phase, an experiment designer recognises the problem and identifies the basic factors required to analyse the problem. The following steps describe the steps the author undertook to identify the factors required for analysis.

1. Selection of quantifying variable

In order to justify the success of any TS FTV or FTM runs, the author must first identify the quantifying variable that needs to be measured.

For the TS FTV cases, the author selects the measure of effectiveness as angle of axial flow deflection measured in degrees in the nozzle outflow divided by the ratio of mass flow injected divided by primary nozzle mass flow. The angle of deflection is obtained by calculating the inverse tangent of the y-component of thrust divided by the x-component of thrust. The measure of effectiveness is given the symbol $\eta_{FTV}$, and this relation in equation format is listed as follows:

$$\eta_{FTV} = \frac{\text{Degrees of thrust deflection}}{\text{Mass flow ratio}} = \tan^{-1}\left(\frac{T_y}{T_x}\right) \div \frac{m_s}{m_p} \tag{1}$$

For TS FTM cases, the author selects the measure of effectiveness as the change in net nozzle thrust divided by the ratio of primary mass flow to secondary injected mass flow. The net nozzle thrust is defined by the following equation:

$$\text{Net Thrust } T_{net} = \dot{m}(v_{exit}) + A_{exit}(p_{exit} - p_{atm}) \tag{2}$$

The first term of the equation, $\dot{m}(v_{exit})$, where the setup is assumed to be a static nozzle experiment, is referred to as the momentum thrust of the nozzle. The second term of the equation, $A_{exit}(p_{exit} - p_{atm})$, is referred to as the pressure thrust of the nozzle.

The net thrust reduction obtained by finding the percentage difference in thrust before and after fluidic injection ($T_0 - T_{inj}$)/$T_0$. The thrust before fluidic injection is calculated from the CFD simulation of the mesh-grid without injection. The measure of effectiveness, $\eta_{FTM}$, is then given by percentage change in thrust divided by the ratio of primary mass flow to secondary injected mass flow

$$\eta_{FTM} = \frac{\text{Net thrust reduction}}{\text{Mass flow ratio}} = \left(\frac{T_0 - T_{inj}}{T_0}\right) \div \frac{m_s}{m_p} \tag{3}$$

2. Selection of factors

The next step to the design of experiments is to select the factors that would affect the quantifying variables defined above. Anthony (2003) recommends the use of a design screening process in order to identify the important variables in the experiment. To perform the analysis, the author used the significant parameters identified in the background literature (Deere et al. 2003) in order to identify the important factors in the experiment.

For both the cases of TS FTV and FTM cases, the author assumes fixed convergent-divergent nozzle geometry. Therefore, the main factors that affect the performance are those that affect the flow either through varying operating conditions or injector geometry.

From the background literature study, both Deere (2003) and Federspiel (1995) highlighted the importance of injector geometry in the nozzle, in the form of injector angle, $\theta_{inj}$, and the slot size, $A_{inj}$ of the injector. The common understanding was that angling the injector to oppose the primary flow, and decreasing its slot size, allowed for better performance using the TS method. The factor of injector angle, defined as the angle between the injector centreline and axial line of symmetry of the nozzle, was chosen for this thesis.
A non-dimensional slot size was chosen by dividing the slot size by the throat area, $A*$, of the nozzle, resulting in a factor defined by the area ratio relationship, $A_{ratio}$.

Also, the previous research determined that the operating conditions of the nozzle have significant effects on the behaviour of the flow field. The main factor that affected the primary flow through the nozzle was determined to be the NPR across the nozzle, and therefore NPR was also selected as a factor.

In the selection of a factor that affected the injector operating conditions, the ratio of injected flow total pressure to primary flow total pressure was selected. Theoretically, this ratio would affect the level of injected jet penetration into the primary flow, which would directly affect the amount of TS of the primary flow. This factor is given the symbol $Pr$.

3. Setting levels of factors

In order to conduct a full factorial study of the experiment, different levels of the factors need to be determined. At the early stages of experimentation, two levels are required for each factor listed above. If a non-linear relationship is expected between the factors and the variables, more levels would have to be defined in order to further study the effects.

The lower level selected for this thesis is based on the factors used in the nozzle that achieved the highest TS FTV in Deere’s paper (Deere 2003). The higher level is obtained by increasing the value of the factors by two. The lower levels are denoted by the symbol $-$, and the higher levels are denoted by the symbol $+$. The factors, and their respective levels used are listed in Table 1 below.

<table>
<thead>
<tr>
<th>Factors</th>
<th>Deere</th>
<th>Lower Level (-)</th>
<th>Higher Level (+)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area ratio, $A_{ratio}$</td>
<td>0.02</td>
<td>0.02</td>
<td>0.04</td>
</tr>
<tr>
<td>Injector angle, $\theta_{inj}$</td>
<td>30</td>
<td>30</td>
<td>50</td>
</tr>
<tr>
<td>Nozzle pressure ratio, NPR</td>
<td>1.5</td>
<td>1.5</td>
<td>3</td>
</tr>
<tr>
<td>Injector-primary mass flow ratio, $Pr$</td>
<td>4</td>
<td>4</td>
<td>8</td>
</tr>
</tbody>
</table>

Table 1. List of factors and levels used

B. Designing phase

In the designing phase, the designer can choose to from a range of design of experiments. The possible designs of experiments range from the varying of one-factors-at-a-time statistical analysis to the conduct of a full factorial experiment. In the consideration of the designs, the author takes into account the size of the experiment based on the number of factors and interactions to be studied.

For a four-factor experiment, the number of interactions, $N$, between the factors is given by the following relationship:

$$N = \frac{n \times (n-1)}{2} = \frac{4 \times 3}{2} = 6$$

(4)

Also, in order to understand the full relationships between the factors, the four factors have to be varied against each other. The number of runs required to conduct a full-factorial experiment on either the TS FTV or FTM case is then given by $2^4 = 16$. For the conduct of both TS FTM and FTV cases, the runs required is then twice that of each experiment, which results in a total of 32 runs.

Within the timeframe of this thesis, a full factorial experiment was reasoned to be achievable, with a strict schedule based on management documentation. A guideline was also required for the runs to be conducted - this is provided through the design of a test matrix describing the interactions. The test matrix used for the experiment is shown in Table 2 on the following page. Each run was conducted in different level combinations of the factors highlighted above. This test matrix setup would allow the author to test the behaviour of all 4 parameters interacting at different levels.
C. Conducting Phase

In the conducting phase of a normal experiment, the experiment designer usually has to discuss the design with the crew for the experiment, and monitor the experiment trials. In this thesis, however, the author is responsible for the conduct of the experiments, as the trials are in the form of CFD simulations of nozzle flow. The following steps describe the process that the author employed for the experiment.

1. CATIA/GAMBIT mesh generation

The first step to any of the runs is to create the basic geometry of the nozzle. The 2D geometry of the TS FTV nozzle is adapted from the nozzle tested by Deere et al. (2005). Deere’s nozzle geometry is a dual throat convergent-divergent nozzle that performed the best TS FTV out of all the experimental studies performed at NASA Langley in the paper.

The nozzle geometry with the best TS performance in the paper is displayed in Figure 10a below. $\theta_1 = 10$ degrees, $\theta_2 = 20$ degrees, $h_{ut} = h_{dt} = 1.15$ inches = 0.02921 m, and $l = 3$ inches = 0.0762 m. The remaining geometry of the nozzle, however, was not specified in the Deere paper. The author had to reverse engineer the geometry of the nozzle by comparing the relative sizes of the remaining dimensions (drawn to scale in Deere’s paper) to the given dimensions. The completed mesh of the nozzle with the nozzle overlaid on top of one of Deere’s simulations is shown in Figure 10b. The overlaid mesh appears to be reasonably close to the geometry of Deere’s nozzle, with the key dimensions of throat and nozzle areas being the same for both cases.

<table>
<thead>
<tr>
<th>Run</th>
<th>$A_{ratio}$</th>
<th>$\theta_{inj}$</th>
<th>$Pr$</th>
<th>NPR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>3</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>4</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>5</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>-</td>
</tr>
<tr>
<td>7</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>-</td>
</tr>
<tr>
<td>9</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>+</td>
</tr>
<tr>
<td>10</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>+</td>
</tr>
<tr>
<td>11</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>+</td>
</tr>
<tr>
<td>12</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>+</td>
</tr>
<tr>
<td>13</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td>14</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td>15</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
<tr>
<td>16</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+</td>
</tr>
</tbody>
</table>

Table 2. Test matrix of thesis
For the purposes of understanding the fundamental fluid mechanics of 2D TS FTV, the nozzle was adapted through the removal of the second convergent region of the nozzle, such that the nozzle expanded into the surrounds directly after diverging from the first upstream nozzle throat. This effectively resulted in the initial converging-diverging nozzle outlet with an outlet height of 0.0473 m, which is almost twice the Deere nozzle’s exit height of 0.02921 m.

Based on Deere’s suggestion of the placement of the boundary conditions, the author placed the boundaries of the environments at 25 cavity lengths downstream, above and below the axial line of symmetry in order to allow space for the nozzle outflow to diffuse to environmental conditions. This geometry was drawn in CATIA v5, a computer-aided drawing software available in the labs. The drawing was then imported to another mesh generating software, GAMBIT, to perform the meshing.

A total of 8 meshes were produced for the test runs in this thesis; 4 meshes were full meshes for TS FTV investigation with injector size and injector angle changes, while the other 4 were half the size of the full meshes for the investigation of TS FTM. An example of the completed mesh generated, both for TS FTV and FTM, is shown in Figure 11 below.

Initially, the author relied on the use of sizing functions to develop the mesh. This was performed by first defining the size of the smallest cells at the edges of the nozzle, then defining the growth rate of the mesh size across the face of the nozzle. This method, however, produced meshes which had too many cells located at the regions of interest, and also tended to produce sudden changes in mesh sizes across the faces of the nozzle to the exhaust region. Over a series of test CFD simulations over several mesh designs, the author recognised that FLUENT is particularly sensitive to these regions where the cell size changes abruptly, or where the cells have a very high aspect ratio, or a high cell length to cell width ratio. In the regions where the cells have the characteristics described, the overall residuals tend to be higher, often causing the corresponding FLUENT simulations to diverge and crash.

After a series of tests using varying meshing methods, the author established that a better alternative to the sizing-function method would be to define the mesh on the edges of the nozzle, and subsequently meshing the face by using the spacing defined by the edge meshes. A rectangular (or ‘quad’ in GAMBIT terms) method of meshing was employed, which used trapezia or rectangles to mesh the face. This method was found to provide
the designer with the best control over the smoothness across the mesh. It also allowed for easier changes to the mesh size where it was required.

In particular, smaller cells were used in areas that require the capturing of higher velocity gradients expected in these regions. For example, significant interaction was expected at the throat, where the flow from the injector was expected to penetrate the primary flow. As shown in Figure 11(b), the mesh developed was finer in the boundary layers along the walls, and also in the throat of the nozzle. There were also requirements to use a finer mesh around the corners at the outlet of the nozzle due to the tendency for the instability of nozzle flow around corners. A grid convergence study was conducted in order to evaluate the sizing of the cells required to obtain a converged solution, which will be discussed later in this thesis.

For the geometry generation of the 2D TS FTM case, the mesh was divided using a line running through the nozzle in the axial direction. This line was defined to be a line of symmetry, and the mesh above the line of symmetry (where no injector was present) was removed. The geometry and mesh that was used in TS FTM is shown in Figure 11(c),(d).

2. FLUENT case setup

The FLUENT software was selected to run the CFD simulations as it has been documented to be capable of making accurate 2D predictions of nozzle flow (Neely, Gesto and Young 2007), and was readily available to the author. FLUENT was installed on 21 computers in the Civil Labs in UNSW@ADFA, where the author conducted all of his simulations.

The 2D double precision (2ddp) solver incorporated in the FLUENT software was used to perform all the CFD simulations in the thesis. The airflow through the nozzle was treated as ideal gas at total temperature of 300 K in order to simulate experimental conditions at sea level room temperature.

Both the primary and secondary flow boundaries were set as pressure inlets, whereas the surrounding boundaries from nozzle exhaust were defined as pressure outlets. Whereas the pressure inlets were varied according to the requirements of the test matrix, the pressure outlets were fixed at 1 atm for all the simulations, thereby simulating atmospheric conditions at sea level.

Initial computational testing performed by the author revealed that the solutions tend to be less steady in the nozzle exhaust due to increased turbulent mixing in the surrounding region. The author had to resort to unsteady solutions in order to predict the flow behaviour as a result of what was understood to be an unsteady state of turbulent mixing at second-order discretisation solvers.

In previous case studies (Deere 2003), however, it was discovered that the presence of co-flow around the nozzle exit managed to reduce the turbulence experienced in this region. The co-flow in Deere’s papers was modelled around the nozzle exit, interacting with the nozzle exhaust. The boundaries of the exhaust directly above and below the nozzle outlet were thus modified to Deere’s values, with the definitions of the boundaries changed from the original ‘wall’ to a ‘mass-flow inlet’ with mass-flux flowing in Mach number of 0.05. The solutions using the co-flow inlet concept are used in the analysis and discussion later in this paper, with majority converging at second-order discretisation. An example of the boundary definitions and their respective locations on the mesh is shown in Figure 12.

At the end of each solver iteration, the residual sum was derived from the conserved variables of the simulation - the variables that determine mass and momentum continuity (e.g. $x$-velocity, $y$-velocity and mass balance). Solution convergence was defined where the average of the scaled residual sum dropping to 3 orders of magnitude below its original values. The main settings and the reasons for their selection in the FLUENT case setup are discussed below.

FLUENT incorporates either a pressure-based or a density-based approach for solving the governing Reynolds-Averaged Navier-Stokes (RANS) equations. According to the FLUENT User Guide 6.3, the pressure-based approach was developed for low-speed incompressible flows, whereas the density-based approach was...
mainly used for high-speed compressible flows. As the flow in the nozzle is expected to be supersonic, the solver based on the density-approach was selected. Steady solutions were generated using first-order numerical iterations followed by second-order iterations.

Based on the density approach, the method for which the governing equations are linearised for conservation of mass and momentum in the flow can be divided into “implicit” or “explicit” forms. The implicit form solves for values using a relation that includes both existing and unknown values from the neighbouring cells, whereas the explicit form solves for values based on a relation that includes only existing values. Both implicit and explicit methods were experimented, with the explicit forms exhibiting more stable behaviour in solution convergence. The method based on implicit forms offered first-order convergence for most solutions, but the residuals de-stabilised in most of the second-order trial runs performed by the author. The method based on explicit-forms managed to converge for second-order solutions, which will be discussed later in the report.

Initial computational testing was done using a one-equation Spalart-Allmaras turbulence model to solve the RANS equations. The one equation model was chosen for its computational efficiency, given the limited hardware available for simulation. The one-equation Spalart-Allmaras model could also resolve steep near-wall gradients, which necessitated the use of a very fine mesh close to the wall (Neely et al. 2007). The solutions obtained by using the Spalart-Allmaras model was then compared to those of the realisable $k-\varepsilon$ 2 equations turbulence model used to solve the RANS equations. The realisable $k-\varepsilon$ model was the preferred model due to three reasons: its capability to accurately model the spreading rate for both planar and round jets in the FTV and FTM nozzles, its superior performance for modelling flows involving boundary layers under strong adverse pressure gradients, separation and recirculation (as defined in FLUENT Guide 6.3) and its capability of modelling more complex features in the flow (Deere 2003). The realisable $k-\varepsilon$ model also includes the latest formulation for turbulent viscosity, and the latest transport equation for the dissipation rate (as of the time of the FLUENT user guide 6.3) to predict turbulence. The term ‘realisable’ represents that the model satisfies mathematical constraints on the Reynolds stresses, and is consistent with the physics of turbulent flows.

The problem with the use of the realisable $k-\varepsilon$ turbulence model, however, lies in the added computational effort required to solve the added equations. This additional time and effort has to be factored into the allocated time for experimentation for the thesis.

3. CFD Verification

The field of CFD studies is largely dependent on its validation potential (Oberkampf and Trucano 2002). It is therefore common practice for verification and validation (V&V) studies need to be conducted to justify the authenticity of the simulation results. In general, the problems of CFD simulations are due to uncertainty and errors, as defined below.

The AIAA Guide defines uncertainty to be ‘the potential deficiency in any phase of activity or activity of the modelling process that is due to the lack of knowledge’ (AIAA 1998). Lack of knowledge is attributed in this case to the complexity of the flow interaction within the nozzle. The lack of knowledge might also include the incomplete characterizations of the parameters such as the turbulence viscosity ratio of the flow boundary conditions. In the understanding of uncertainty, a sensitivity analysis is performed by varying a component of the model, such as an input parameter and measuring the effect of the variation of the model on the output quantities. For FLUENT's density-based solver, the main control over the time-stepping scheme is given by the Courant-Friedrichs-Lewy number (CFL); a sensitivity analysis is conducted on the CFL number by varying the CFL number and measuring its effects on the output quantities.

The AIAA Guide defines error to be ‘a recognizable deficiency in any phase or activity of modelling and simulation that is not due to lack of knowledge’ (AIAA 1998). This definition of error dictates that error is divided into acknowledged or unacknowledged. Examples of acknowledged errors include round-off error in a digital computer and physical approximations made to simplify the modelling of a physical process. Unacknowledged errors refer to mistakes by the designer or the experimenter (Stern et al. 1999).

Four dominant errors are usually associated with the running of CFD simulations, namely insufficient spatial discretisation convergence, insufficient temporal discretisation convergence, lack of iterative convergence, and computer programming errors. The objective of verification study is to estimate the discretisation error of the numerical solution.

Verification is defined as the process of determining if the computational simulations accurately represent the conceptual model of the designer, without comparing the results to the real world behaviour. The stage at which the verification process fits in the design process is as shown in Figure 13 on the following page.
A grid-dependence study is conducted to investigate the dependence of important variables on spatial and temporal discretisation in the mesh. The grid size of the mesh is steadily decreased across three different mesh sizes. Theoretically, as the grid size and time step approaches zero, the discretisation error asymptotically approaches zero. Richardson’s extrapolation is applied on the results obtained from two finest grids to approximate a ‘true value’ for the result. The result from the coarser mesh is then compared to this extrapolated result, which gives an indication of error in the discretisation process.

4. CFD Validation

Validation is the process of determining if a computational simulation represents the real world. The real world is best reflected in the form of experimentation data, even though experimentations may not be as accurate as computation simulations (Oberkampf and Trucano 2002). The only previous conduct of TS FTM experiments by Federspiel provided an example of CFD validation capabilities (Federspiel et al. 1995); however due to the difference in nozzle designs the results of Federspiel’s experiments could not be directly validated against the simulations run in this experiment.

As the author was learning to understand the use of FLUENT CFD method at the beginning of the thesis, the author conducted a basic validation study of the meshing and FLUENT case setup based on the nozzle designed in Deere’s paper (Deere 2003), with a geometry described in the sections above. Deere performed extensive experimentation on a wide range of nozzles, and published results for both SVC and TS FTV. For the purposes of enhancing understanding, the author attempted to reproduce the TS FTV case for which Deere established the most significant FTV angle for percentage secondary mass flow injected.

The process of reproduction and validation was designed to help the author understand the important parameters affecting FLUENT case setups, and also provide justification in the methods used by the author for creating and meshing geometries. It also provided the author with validation that FLUENT was capable of reproducing the lab results, given the same boundary conditions and mesh-grid geometries, as the PAB3D software that Deere employed. The results from these validation attempts by the author are described in the ‘Initial Computational Testing’ section in this thesis.

Another potential for validation of the CFD simulations lies in the adaptation of the UNSW@ADFA FTV setup. The UNSW@ADFA experiment setup is set for SVC FTV experiments, chosen for ease of visualisation using a schlieren optical

![Figure 13. Verification process (AIAA 1998)](image)

![Figure 14. Validation Process (AIAA 1998)](image)

![Figure 15. Setup of UNSW@ADFA SVC FTV experiment (Stagg 2008)](image)
system. In the setup, four gas bottles filled with 20 MPa compressed air provide the primary flow to the nozzle, while a 20 MPa air bottle was used to supply the secondary flow (typically operated in the range of 1 to 7 bar), as shown in Figure 15 on the previous page.

More importantly, Stagg also changed the FTV nozzle design to facilitate changes in nozzle geometry by making the nozzle easily removable. The current nozzle design in the UNSW@ADFA test rig is a conventional nozzle design as shown in Figure 16 with dimensions of \( w_t = 3 \text{ mm}, A^* = 30 \text{ mm}^2, \text{AR} = 3.33, w_e = 10 \text{ mm}, A_e = 100 \text{ mm}^2 \) and \( \text{AR} = A_e/A^* = 3.33 \). The value of the AR of the nozzle was specifically selected such that the flow structure of 2D numerical simulations attained would be similar to 3D numerical simulations. In the initial computational testing phase the author also attempted to replicate the computational simulations as part of the FLUENT learning process, to limited results.

As the nozzle geometry is designed to be removable and interchangeable, potential lies in the production of a nozzle piece with the optimum TS FTV and FTM geometry proposed in this thesis, and comparing it to the results of the CFD simulations performed. Due to timing constraints, however, this validation process could not be completed within the thesis timeframe; future work into TS FTV and FTM research should take into account the validation possibilities of modifying the UNSW@ADFA nozzle.

D. Analysis Phase

The next phase of the design is to analyse and interpret the results from the experiment. The aim of the analysis phase is to determine the design parameters that affect performance and variability. The results are also interpreted to identify the design parameter levels that yield the optimum performance. At the end of the analysis phase, required further improvements are highlighted. Three ‘simple’ tools are recommended by Anthony to facilitate the analysis of experiment results.

1. Main effect plots

Main effect plots are the conventional tool for analysis of data. The plot relates the mean response values of the variables to the level of the design factors, and as such the relative strengths of the effects of the factors can be obtained. For analysis, the effect of a design factor, \( E_f \), is defined to be the difference between the mean response values (\( \eta_{FTV} \) or \( \eta_{FTM} \)) when a particular factor is changed from high (+) to low (-) levels.

The sign (positive or negative) of \( E_f \) will signify the relationship of the factor with the response values – a positive \( E_f \) implies that increasing the factor increases the response values. The magnitude of \( E_f \) directly reflects the relative strength of the effect of the factor on the response values.

For the understanding of the effects of two-factor interactions, the effect parameter \( E_f \) is determined by measuring the difference of the mean response values between when both factors are at high levels and when both factors are at low levels. The signs and magnitudes of \( E_f \) then have the same implications (this time for two-factor interactions) as described above.

The main effects analysis is performed for this thesis by using the average of the quantifiable variables \( \eta_{FTV} \) and \( \eta_{FTM} \) as the main response values for the analysis of TS FTV and FTM respectively.

2. Interaction plots

Interaction plots will be used to identify the relationship between different factors. An example of an interaction plot is shown in Figure 17 (Anthony 2003). The purpose of the interaction plot is a graphical tool which plots the interaction of the two factors under consideration at all possible combinations of their levels, and shows the relationship of varying the factors on each other.

At each combination of factors (-1 being low level and +1 being high level), the mean response value is obtained from the FLUENT simulations. The lines are then drawn joining the two points caused by setting a factor at high level. For example, in Figure 17 the points for which the factor A is set at high level are joined, and the same is done for when the factor A is set at low levels. The gradient of these lines then reflect the effect of changing the level of B with level A fixed on the mean response values. A positive gradient implies that increasing the level of B would increase the mean response values.
For factors that do not have significant interactions, the two lines would be parallel, implying that fixing the factor A at different levels does not significantly affect how the changes in the levels of B affects the mean response value. If the factors are significantly non-parallel (as seen in the example figure), however, a significant interaction between the factors is expected. For example, in Figure 17, when the factor A is set at a high level (+1), the change in the level of factor B does not have a significant effect on the mean response values. In contrast, when the factor A is set at a low level (-1), changing the level of factor B causes a significant change in the mean response. It can therefore be concluded that there is a significant interaction between the factors of A and B, which needs to be reviewed in the determination of significant parameters in the nozzle geometry.

3. Pareto plots

Pareto plots are plots that are used to detect the level of significance of factors and their mutual interactions. The standardised mean response values of the parameters in a bar chart, where the relative heights of the bars are compared and significant height differences between the bars are identified. An example of a Pareto plot is shown in Figure 18. A reference line (dotted) is drawn where results are determined to be significant, usually where there is a significant change in the length of the bars on the Pareto plot. Any factor that has a bar extending beyond this line is potentially significant to the experiment, and therefore needs to be reviewed in detail.

For this thesis, the standardised values are obtained through the division of the values of \( E_f \) by the average value of \( E_f \) obtained from the experiments. As per the example, a reference line is drawn where the height of the bars changes significantly, and any factor which extends beyond this line is determined to be significant.

VI. Initial Computational Testing

As previously described, initial computational testing was conducted in order to verify and validate the CFD model used by the author. The purpose of initial computational testing was to help the author understand the method and process required for producing valid CFD simulations. The setup of the processes are such that they facilitate the development of the author’s knowledge in the production of CFD simulations, and provide validation for the CFD method. Initial computational testing also allowed the author room to experiment with the selection of important parameters for the geometry, such as the CFL.

A. Mesh grid dependence study

The author generated a total of 8 meshes for the purposes of testing TS FTV and FTM. A mesh dependence study was conducted for the basic mesh that all the other 8 meshes were derived from. This geometry was a FTM simulation with \( A_{ratio} = 0.02, \theta_{inj} = 30 \) degrees, \( Pr = 1.5 \) and NPR = 4.
The simulations running on other meshes were assumed to have similar mesh dependencies, as the main nozzle geometry was similar, with only slight changes in the angle of injection and injection size. For the FTV cases, the mesh-grid in the nozzle was reflected about the axial line of symmetry of the nozzle to produce the full mesh. Three different sizes of mesh sizes were then generated, with the mesh size been refined by a grid refinement ratio, \( r_G \), of approximately 1.9 from the coarsest grid in order to avoid a known issue of interpolation errors from using other \( r_G \) values (Stern et al. 1999). Reading from the FLUENT case file, the coarsest mesh-grid had 63480 cells, the intermediate mesh-grid had 121085 cells and the finest mesh-grid had 230060 cells. The first two (coarser) meshes managed to converge with second-order discretised solvers. The finest mesh took a significantly longer time period to converge with even the first-order discretised solvers, and its residuals exhibited oscillatory behaviour with second-order discretised solvers. The problem of convergence with the finest mesh could be attributed to the computer hardware having problems handling the calculations required per iteration for the increased number of cells, compounded by the large fluctuations in the flow due to complexities of fluid injection at the throat of the nozzle.

Comparing the solutions from first-order convergence, the coarsest mesh obtained a \( \eta_{FTM} \) value of 2.67, whereas the intermediate and finest mesh had a \( \eta_{FTM} \) value of 2.262 and 2.278 respectively.

In order to identify convergence conditions, the solution changes, \( S \), were defined to be the difference between the results obtained from grid size changes. Therefore, for the results obtained from coarse to intermediate to fine, \( \epsilon_{coarse-int} = |2.67 - 2.262| = 0.408 \), and \( \epsilon_{int-fine} = |2.262 - 2.278| = 0.016 \) respectively. The solution change ratio, \( r_s \) is then given by the ratio of \( \epsilon_{int-fine} \) to \( \epsilon_{coarse-int} \), which is 0.016/0.408 = 0.0392. The value of \( r_s \) is between the range of 0 and 1, which places the convergence conditions in the range of monotonic convergence (Stern, et al. 1999). The generalised Richardson-Extrapolation (RE) can be used in this range to estimate errors and uncertainties of the condition.

Assuming that the functions used in the grid and the order-of-accuracy, \( p_k \) does not change with step size, Stern quoted that the order-of-accuracy can be obtained by the following relationship:

\[
p_k = \frac{\ln(\epsilon_{coarse-int} - \epsilon_{int-fine})}{\ln(r_s)}
\]

For the simulations that were performed, \( p_k = -1 \), indicating that the order of accuracy is the same as the (first-order) discretisations used for obtaining the solutions, which is reasonable. The uncertainty in the code, \( U_k \) is then estimated for the finest solution in the code based on another relationship quoted by Stern:

\[
U_k = \frac{\epsilon_{int-fine}}{r_k^{p_k} - 1}
\]

Substituting the values obtained previously, the uncertainty estimated based on fine-grid simulations, \( U_k \), is approximately 0.000653, which is approximately 0.01%. This is a very small uncertainty, which could possibly be attributed to the fact that even the coarsest grids used in this study had a significant number of cells. Using the solution change \( \epsilon_{coarse-int} \) instead of \( \epsilon_{int-fine} \) in the above relation for calculating uncertainty, the uncertainty of estimations was calculated to be 0.01666, which is approximately 2%. This is still within a 5% acceptable or allowed range for solution uncertainty expected by the author. Therefore, the coarsest mesh-grid produced in this study was used for the test matrix computations.

B. Courant number (CFL) dependence study

In order to conduct an understanding of the effect of the CFL number on the simulation results, simulations had to be run with different CFL number. As the CFL determines the temporal and spatial sizing of the simulations, lowering the CFL is expected to produce more accurate results. In terms of computational effort required, having a lower CFL would generally increase the CFD simulation time period required, due to the decreased in time-step size per iteration. Unfortunately, the author did not measure the time required for the solutions to converge, thus the factor of time and effort efficiency could not be completely quantified.

The mesh used for the mesh dependence study was re-used for this study, with the same factors as previously highlighted. The CFL was changed between 0.25, 0.5 and 0.75 (increments of 0.25) in order to investigate the effect of CFL on the results.

For convergence at second-order discretisation, the \( \eta_{FTM} \) values obtained were 2.96, 2.90 and 2.84 for the CFL values of 0.25, 0.5 and 0.75 respectively. This reflected a reverse relationship between CFL and the value of \( \eta_{FTM} \) obtained.
A plot showing the effects of changing the Courant number on the $\eta_{FTM}$ values is shown in Figure 19. As the CFL number is reduced, the value of $\eta_{FTM}$ appears to approach an asymptotic value of 3. A change of the CFL number from 0.75 to 0.25 (67% decrease) induced only a 4% increase in the $\eta_{FTM}$ values. Further systematic decreases in CFL would be expected to fall closer to the $\eta_{FTM}$ value of 3; this relationship would be better investigated with the conduct of more experiments with lower CFL numbers (to obtain more grid points for the curve). Eventually, the $\eta_{FTM}$ value should be independent of lowering of CFL numbers.

In the selection of the CFL number to use in the simulations, it is worth noting that more volatile interactions would be expected than the ones present in the basic factor levels selected. With increases in NPR and $Pr$, more jet penetration was expected and hence more turbulence and high-shear fluid interactions were expected in the nozzle. The use of larger CFL numbers could therefore potentially lead to divergence of the FLUENT simulations. The CFL value of 0.25 was therefore selected in order to increase the accuracy of the solution, and also to minimise possibilities of divergence in the simulations.

C. Validation of results against Deere Nozzle

As previously discussed in the CFD setup section, the author attempted to validate his thesis methods by using the Deere nozzle geometry. The focus of this validation process was to be able to identify the key flow behaviour compared to the Deere nozzle, and also to be able to obtain the same value of $\eta_{FTV}$ as that of Deere. From initial runs, the results from the simulations were established at first-order convergence, with residuals converging to $1/1000^{th}$ of their initial values.

At second-order discretisations, however, there appeared to be considerable turbulence in the flow, especially in the areas of the throat and the nozzle exhaust. Convergence was defined where the residuals and other important parameters (like mass flow rate) exhibited signs of asymptotic-behaviour. The second-order converged solutions is as shown in Figure 20(a). The white blotches missing from the nozzle flow region are out-of-range because the author scaled the range of Mach number in the display to fit Deere’s scale of Mach numbers. It is important to note that none of the turbulence models or boundary conditions has been modified in the switch between second-order discretisation and first-order discretisation methods.

![Figure 19. Effect of changing CFL](image1.png)

![Figure 20a. Mach-number plot of solution at second-order iterations](image2.png)

![Figure 20b. Mach-number plot of Deere’s solution (Deere et al. 2005)](image3.png)
Figure 20(b) shows the plot of the Mach number distribution in Deere’s paper (Deere et al. 2005). In contrast, Deere’s solution appears to be more realistic than the second-order solution achieved by the model used in this paper.

The instability of the flow in Figure 20(a) is most significant with complex Mach number changes in the nozzle right after the injection at the throat, and also significant turbulence at the exhaust of the nozzle. From the large amount of instability in the flow it can be deduced that the injection in the throat induced significant oscillations in the exit flow of the nozzle, and therefore there was a possibility that the flow in the region could be unsteady. The flow field that Deere simulated in Figure 20(b) was achieved using unsteady solvers using the \( k-\varepsilon \) model for turbulence; the flow field exhibited no signs of instability of Mach number interactions, and reflected a consistent band of high Mach number across the middle of the nozzle.

In order to identify the difference between steady and unsteady flows, the results obtained from using a steady explicit solver in FLUENT was compared to that of an unsteady explicit solver with the same boundary conditions. The results of first-order converged solutions were also compared to the results of second-order simulations where residual values show signs of persisting oscillations around a certain value. Eventually, the co-flow boundary condition was added into the system (as discussed previously) in attempts to stabilise the diffusion of the turbulence in the nozzle exhaust regions. The method of co-flow addition was identified to be the best method for valid results, and as such for the simulations in the test matrix the addition of co-flow was the preferred method.

The following sections summarise the lessons learnt by the author through the attempts to validate the results of the simulations against Deere’s nozzle.

1. First-order and second-order convergence comparison

   The difference between first and second-order convergence lies fundamentally in the method used to discretise the RANS equations. By definition, the first-order discretisation method omits terms second order or higher in a Taylor series expansion, and is therefore potentially one order less accurate than the second-order discretisation method. In the actual CFD simulations, however, increasing the order of the method of discretisation may not significantly affect results. The author therefore investigated the difference between the results at first-order convergence and second-order simulations. The result for the Deere nozzle using a first-order solver is shown in Figure 21. The second-order solution was plotted previously in Figure 20(a).

   In first order simulations, the contours of Mach number are smoother than that of the second-order converged solutions. This is to be expected since the first-order solvers do not account for second-order interactions in the flow, such as the solution and the gradients of Mach number are less susceptible to oscillations. The Mach number distribution of the first-order solution is more realistic and closer to Deere’s solutions without the severely unstable mixing in the second-order solution.

   There is also less diffusion of the nozzle exhaust in the first-order solutions compared to the secondary flow. Both plots use the same scale, and the direct comparison of the lowest contour with value of 0.08 is significantly more spread out in the second-order case. This could again be attributed to the lack of turbulent mixing in the first-order case, which potentially diffuses the jet further beyond normal convective activity in the exhaust.

   Worth particular note is the relative steadiness of flow activity within the nozzle, in particular around the throat. Whereas the second-order case exhibits signs of turbulent flow starting around the region of injection, the first-order case reflects a significantly smoother mixing of the jet injection into the primary flow. The separation of the subsequent flow from the nozzle lower wall is marked by a distinct region of lower Mach numbers (a blue band) before there is more mixing at higher speeds around the corner where the nozzle re-converges.

   Although the first-order solution exhibits a flow-field much closer to the Deere’s model, there is also a possibility that the flow-field exhibits more turbulent mixing than is modelled by the first-order solutions in
real-flow environments, especially in the interactions between jets at the throat of a nozzle. The extent of turbulent mixing in the nozzle exhaust beyond that shown in Figure 20(b) is not discussed or displayed in Deere’s papers (Deere’s grid did not pay attention to far field behaviour), and therefore there is no data to validate either of the far-field mixing behaviour shown in the plots. Deere’s results, however, were second-order CFD unsteady solutions, which suggest that further work in the refinement of the grid and boundary conditions could be performed in the current model.

The calculations of the $\eta_{FTV}$ for the first-order and second-order cases are 4.96 degrees/%-injection and 14.09 degrees/%-injection respectively. Given that Deere’s analysis obtained a $\eta_{FTV}$ value of 3.9 degrees/%-injection, the difference in the second-order case becomes significant. The over-performance of the TS FTV nozzle can be attributed to high-speed turbulent mixing in the exit plane of the nozzle, where the x and y components of thrust are measured. Although the first-order case is closer in performance to the Deere’s nozzle, there still exists a significant difference of the magnitude of approximately 27%. The unsteady case should therefore be investigated in order to understand the behaviour of the nozzle over time steps.

2. Steady and unsteady solutions comparison

Whereas in steady solutions the flow behaviour is fixed over time, the unsteady solver evaluates the RANS equations through the use of time-step iterations. For the unsteady test case in this thesis, the author used the CFL of 0.25 to define the explicit step sizes required. At second-order iterations, the unsteady solutions showed signs of asymptotic oscillatory behaviour of mass-flow rate values and the exit velocities of the nozzle. A plot of the Mach number distribution in the flow-field is shown in Figure 22 below.

The flow dispersion is more gradual in the unsteady case, and the second-order simulations do not exhibit as much turbulent behaviour in the throat as the steady second-order simulations. The spread of the exhaust flow is more realistic in that the spread of the flow is neither highly turbulent like the second-order steady case, nor is it slow and almost non-developed in the first-order case. The problem, however, lies in the fact that the turbulent flow in the exhaust field appears to develop an oscillatory behaviour in the nozzle exhaust field. The continuous shifting of the nozzle flow in this region would cause the residuals to also exhibit an oscillatory behaviour, and cause a fluctuation in the dispersion and diffusion of the flow. Because of this oscillatory behaviour, the values of the effectiveness parameter $\eta_{FTV}$ calculated changes over time. The value of the parameter therefore needs to be averaged over a time span for a single oscillation of the diffusing wave.

The FTV nozzle performance parameter $\eta_{FTV}$ in the unsteady second-order simulation is 4.03 degrees/%-injection, which is only a 3% error compared to the result of 3.9 degrees/%-injection in Deere’s paper. There is therefore evidence to conclude that the unsteady second-order case has the most validity out of all the simulations the author ran.

The flow behaviour between the Deere nozzle and the unsteady second-order computational simulation run by the author is compared in Figure 23 below.
The plot of the results in the simulation was deliberately scaled to match that of Deere’s, which resulted in patches of white spots in the figure where regions were above or below this range of values. The problem with graphical comparison lies in the problem of the difference in the level of zoom, and the region of zoom of the figures.

The comparison of the Mach-number plots of the nozzles shows that the primary flow of the nozzle from the simulations (the green band in the middle of the nozzle) exhibits a similar behaviour to that of Deere’s nozzle. The average Mach number of the primary flow is in the range of Mach 0.9-1.1.

There are, however, several differences in the Mach number distribution in the recessed cavity and in the nozzle exhaust. The proportion of the flow that reaches the Mach number of 1.8 is significantly higher in the author’s simulation, although both patches of high Mach numbers are similarly shaped.

There is also a bigger proportion of the flow that reaches high Mach number at the top corner of the nozzle throat. Whereas the Mach number barely reaches 1.8 in Deere’s nozzle, from the simulation the author ran there was a significant region of high Mach numbers with the contour of 1.8. This difference can be attributed to the difference in the way the author designed the corner of the nozzle throat compared to Deere. Deere employed the geometry of a smooth throat corner around the throat (and also at the converging-diverging section in the dual throat), whereas the author used angled straight lines to form a ‘flattened’ throat instead of a sharp corner or a curved surface at the throat. Although this difference affects the way the flow turns around the throat corners, the general flow behaviour is not severely affected, and the trends analysis can still be conducted using the general behaviour of the flow.

VII. Results Discussion

The tables with the values recorded from FLUENT and the calculations required can be found in Appendix E. The author adopted an approach of using steady first-order explicit solvers with CFL = 0.25 to obtain an initial overview of the flow in nozzles. Upon achieving first-order convergence of residuals, the author switched to second-order steady explicit solvers to obtain steady second-order convergence in the residuals. If the residuals exhibited oscillatory behaviour, the author used dynamic grid adaptation every 1000 iterations to refine the grids with a refine threshold of 5% of the maximum Mach number value. If the residuals still diverged, often causing a floating point error in FLUENT, the author would change the solvers to unsteady solvers.

This approach had the disadvantage that if the flow simulations were unsteady in nature, like the Deere nozzle, the time spent on the steady solver would not produce accurate results. The approach, however, would allow the author to produce most of the simulations in the limited amount of time available, which was required for the analysis to follow. In the event of the requirement for an unsteady solver, the author would at least have a converged first-order simulation to reference general trends of data.
A. General results overview

The average number of iterations required for TS FTM cases was in the range of 10000 to 20000 iterations to converge with second-order discretisation. This translates to approximately 17 hours of simulation time required per TS FTM simulation. Out of these TS FTM simulations, only one case failed to converge to steady second-order discretisation, run 16 - where all the factors were set at high levels. In comparison, for the TS FTV cases, half of the cases failed to converge to steady second-order discretisation. Only the cases with the $A_{ratio}$ factor set at lower levels (0.02) managed to obtain steady second-order convergence. Unsteady analysis was therefore conducted on the remaining cases. A summary of the level of convergence achieved by the author is in Table 3 below, with ‘-’ signifying low level of the factor and ‘+’ signifying high level of the factor in the setup.

All the nozzle flow fields can be found in the Appendix F. The following sections will discuss the highest performing nozzle geometry from both TS FTV and TS FTM cases. The focus of the thesis, however, lies in the discussion of the interaction between factors, which will be discussed in more detail.

B. TS FTM Results Discussion

All of the TS FTM simulations were performed with the symmetry line modelled as a 2D line of symmetry. This meant that the model was effectively reflected on the line of symmetry, and extruded into unit depth for all thrust calculations, and the throat had a rectangular cross-section. The injector was also then modelled as a slot, with the injection area, or slot size, used in the factor $A_{ratio}$. Although the modelling of 2D geometry is appropriate for potential validation in the UNSW@ADFA setup, the ‘real world’ geometry would be in 3D, and 3D modelling would be required.

1. Axisymmetric and 2D symmetry comparison

FLUENT provides an alternative to actually creating a 3D mesh through the use of an axisymmetric definition at the line of symmetry. This means that the model is revolved around the line of symmetry instead of reflected and extruded. The nozzle throat cross section in the axisymmetric case is circular, and the injector is modelled as a circular slot around the circumference of the throat. In order to understand the difference, a short axisymmetric to 2D results comparison was performed on the case that had the $\eta_{FTM}$ value closest to the mean $\eta_{FTM}$ value obtained over the entire study. This case was run 11 with settings as described in Table 3.

<table>
<thead>
<tr>
<th>Runs</th>
<th>$A_{inj}$</th>
<th>$\theta$</th>
<th>Pr</th>
<th>NPR</th>
<th>TS FTM Steady order of convergence</th>
<th>TS FTV Steady order of convergence</th>
<th>TS FTV Unsteady order of convergence</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>2</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>3</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>4</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>5</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>7</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>8</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>9</td>
<td>-</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>10</td>
<td>+</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>11</td>
<td>-</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>12</td>
<td>+</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>13</td>
<td>-</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>14</td>
<td>+</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>2</td>
<td>1</td>
<td>1</td>
</tr>
<tr>
<td>15</td>
<td>-</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>2</td>
<td>2</td>
<td>-</td>
</tr>
<tr>
<td>16</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>+</td>
<td>1</td>
<td>1</td>
<td>1</td>
</tr>
</tbody>
</table>

Table 3. Summary table of order of convergence achieved in CFD simulations

Final Thesis Report 2009, UNSW@ADFA
A comparison of the Mach number distribution in the flow field is provided in Figure 24, with the axisymmetric case in the top half and the 2D planar case in the bottom half.

With the Mach numbers scaled on the same scale, the axisymmetric case in Figure 24 achieves higher Mach numbers in the middle of nozzle, achieving a maximum value of approximately 3.63 compared to a maximum of about 2.50 for the 2D TS FTM case. The development of Mach-disk-like flow can also be identified in the case of the axisymmetric model. The injection of fluid at the throat appears to cause the primary flow to separate from the nozzle walls, and as the flow accelerates to supersonic speeds the expansion behaviour of the flow turns the flow back into the wall, and a strong compression wave is formed at the wall, merging together to form an oblique shock. The interaction of the oblique shocks in the middle of the nozzle causes the formation of a Mach disk, or a normal shock in the middle of the nozzle. Flow across the normal shock increases in pressure and decreases in flow velocity, the latter of which is especially desired in the case of TS FTM.

This flow behaviour is not observed in the 2D TS FTM model, where the formation of compression waves appears to interact almost exactly on the nozzle centreline due to the symmetry of the flow conditions. In contrast to the axisymmetric case, the flow of the planar 2D case does not appear to develop strong oblique shocks at the nozzle exit. Only a faint sign (partially due to the effects of scaling) of a compression wave can be noticed at the nozzle exit in the 2D TS FTM case, with the wave interactions between expansion fans and compression waves repeatedly interacting within the Mach envelope, in what appears to be a shock-diamond structure which is typical of over-expanded flows.

Calculations regarding the flow parameters of the nozzle showed that the pressure thrust in the axisymmetric case was greater than that of the planar 2D case, the latter of which actually encounters pressure drag. The momentum thrust in the axisymmetric case, however, was significantly smaller than the planar 2D case. Whereas the planar 2D case generated a momentum thrust of about 12709 N, the axisymmetric case only induced a momentum thrust of 590 N. This significant drop in thrust in the axisymmetric case is attributed to the increased mass-flow of injection as the injector slot area is a ring around the circumference of the nozzle at the throat, therefore injecting significantly more mass into the nozzle compared to the planar 2D case, where a single slot of unit depth is modelled on either side of the throat. The increased injection causes the primary flow to be squeezed to a smaller cross sectional area, and therefore mass-flow rate through the nozzle is restricted, and thrust is reduced. The axisymmetric case achieved a $\eta_{FTM}$ value of 9.15, significantly greater than the value of 1.54 achieved in the 2D case.

In the design of a real-flow nozzle, however, it is not practical for the injector slot to cover the entire throat – it is more practical to cut slots into the existing throats, at intervals with each other, and inject only in the slots required to perform either FTM or FTV operations. The use of slots instead of a ring-injection would be expected to decrease the level of thrust modulated. As such, there is a performance compromise required in the design of the slots, which should be reviewed by the designer when selecting the number of slots required to achieve the level of performance desired.

From this study, there is much potential identified for the development of the asymmetric, or close-to asymmetric TS FTM nozzle. Because of time limitations, however, it was reasoned that it would be useful and practical to complete full planar 2D case studies to understand the effects of slots on the FTV or FTM system. For the purposes of the study of effects of varying parameters, a planar 2D trend study would prove sufficient, as is reflected by similar compression-wave and oblique shock structures behaviour in the two flows. The intrinsic details of the flow, however, can only be investigated by performing a similar study with asymmetric models in the future.
2. Best-performance flow-field analysis

As listed previously, most of the TS FTM cases converged to second-order discretisation levels. The only exception to this geometry is run 16, with all of the factors set at highest levels. The average $\eta_{FTM}$ level achieved is 1.66 % injection. The highest $\eta_{FTM}$, with a value of approximately 2.96 % injection, was achieved in run number 1 - with all the factors set at low levels. The lowest $\eta_{FTM}$, with a value of 0.78 % injection, was achieved in run number 16, with all the factors set at high levels. This intuitively reflects that the factors have an inverse relationship with $\eta_{FTM}$ – the higher the levels are, the lower the achievable FTM.

The nozzle flow field for the highest-performing case, run 1, is in Figure 25. As per the average case discussed previously, the injection of fluid at the throat causes separation of the primary flow from the nozzle wall at the throat of the nozzle. The injection of fluid induces a high velocity shear layer in the region of the throat, and effectively squeezes the primary flow to a smaller throat cross-section, as is reflected by the pathlines of fluid particles in from the throat. Assuming that the original nozzle flow was choked at the throat, the speed of the flow is fixed at $M = 1$ at the throat. Assuming no changes in temperature, this decrease in throat cross section then effectively decreases mass flow rate; since mass flow rate is equal to the product of density, velocity and throat cross-sectional area.

The injection of fluid at the throat also induces a second effect on the flow in throat – the primary flow that was turned away from the throat by the injection fluid then begins to accelerate and ‘spread’ supersonically, effectively turning the combined primary and secondary flows back towards the nozzle wall. When the combined flow encounters the nozzle wall, a strong compression wave is generated, forming an oblique shock at the point where the combined flow meets the wall again. This oblique shock turns the flow parallel to the nozzle wall, where the flow is accelerated through the diverging portion of the nozzle. The oblique shock interactions meet at the centreline of the nozzle with its counterpart (in a 2D simulation the nozzle geometry is reflected), and the shock-shock interactions forms a diamond shape, similar to the shock diamonds typically found downstream of slightly under-expanded or over-expanded flow. The interactions of the oblique shocks also serve to increase the pressure and decrease the velocity of the flow, although not as much as the Mach disks (normal shocks) in the flow.

Since the area ratio for nozzle-exit to nozzle-throat is fixed at $A_{exit}/A^* = 1.619$, the exit Mach number is determined through compressible flow relations by the following relationship, where $\gamma = 1.4$ for ideal gas:

$$
\frac{A_{exit}}{A^*} = \left(\frac{\gamma + 1}{\gamma - 1}\right)^{\frac{\gamma + 1}{\gamma - 1}} \left(1 + \frac{\gamma - 1}{2} M_e^2\right)^{\frac{\gamma - 1}{\gamma + 1}}
$$

Solving the equation for $M_e$ gives the value of $M_e = 1.949$ at the nozzle exit. The ratio of the $p_{exit}$ to $p_t$ is obtained by the following relationship:

$$
\frac{p_{exit}}{p_t} = \left(1 + \frac{\gamma - 1}{2} M_e^2\right)^{\frac{\gamma - 1}{\gamma + 1}}
$$

This ratio was calculated to be 0.1381. Since $p_t$ is 4 atm, $p_{exit}$ is then obtained to be 0.55 atm, which is below the atmospheric value of 1 atm (note: units of pressure set as atm instead of Pa for ease of understanding). This implies that with the basic converging-diverging nozzle geometry, the nozzle is over-expanding. Since the squeezing of the throat by the secondary injection effectively decreases $A^*$, the nozzle area-ratio as defined in Equation 7 increases. With increased nozzle area-ratio, the flow is expanded further, hence the $M_e$ increases. The increase in $M_e$ then implies that the pressure ratio as described in Equation 8 decreases. This implies that
\( p_{\text{exit}} \) is further lowered across the expansion, resulting in further over-expansion of the flow. This explains the formation of another set of oblique shocks was formed at the exit plane of the nozzle.

The flow structure obtained through all the simulations are also similar to the formation of Mach disks and shock diamonds – the set of oblique shocks at the nozzle exit appear to interact to form a slight normal shock, a Mach disk, slightly downstream of the shock. The envelope around the shock interactions at \( M = 1 \) also exhibits barrel-like behaviour expected of envelopes surrounding shock diamonds. A hint of a disturbance caused by the weakening oblique shocks formed in the mid-nozzle section appears to disturb the Mach envelope slightly downstream, but this disturbance is not expected to have significant effects on the flow.

The flow field as discussed above is consistent with conventional understanding of nozzles, and can be explained through a series of shock-wave interactions. It is worth considering, however, the effects that modelling the flow in 2D has on the flow field. The primary concern in this case is the lack of representation for the possibility of primary jet flow turning around the injected fluid. In a 2D case, the injected flow is similar to a significant obstruction in the nozzle that causes the flow to turn away from its throat, but in 3D cases the level of penetration of the secondary flow might vary as the primary flow could manoeuvre and mix around the secondary injection. The axisymmetric case study has also shown that the interaction of the oblique shocks is significantly different in a 3D environment. Therefore, for future work, studies into 3D jet penetration and associated TS FTM should be considered in more detail.

This data is used for the following analysis, based on the assumption that the performance of TS FTM in 2D exhibits similar trends with changes in the factors as an axisymmetric or 3D environment.

3. Interaction plots

For a more detailed analysis, interaction plots were produced in Figure 26 below.

![Interaction plots for TS FTM](image)

[All y-axis are values of \( \eta_{\text{FTM}} \)]

The interaction plots are displayed in a matrix format, where the first row relates to comparisons with area ratio \( A_{\text{ratio}} \), second row (and first column) relates to comparisons with \( \theta \), third row (and second column) relates to comparisons with injection ratio \( Pr \), and the third column relates to comparisons against \( NPR \). The blue lines with diamond-shaped end points relate to the variations of \( \eta_{\text{FTM}} \) with fixed low levels of the factor designated in the rows against changes of the factor listed in the columns. The red line with square-shaped end points relate to the variations in \( \eta_{\text{FTM}} \) with fixed high levels of the factor designated in the row against changes of the factor listed in the columns. For example, the blue line with diamond-shaped end points in the first row and first column reflects the changes in \( \eta_{\text{FTM}} \) with \( A_{\text{ratio}} \) fixed and \( \theta \) varied from low to high levels. All the plots are plotted on the set axis of -1.5 to 1.5 in the x-axis, and 1 to 2.5 in the y-axis (values of \( \eta_{\text{FTM}} / \% \)-injection).

The gradient of each individual line plotted reflects the effect of varying the factor in the x-axis from low to high levels on \( \eta_{\text{FTM}} \), with the other factor (being compared against) being held constant. The steeper the gradient
of the line, the more the $\eta_{FTM}$ value is affected by changes in the factor. From the plots, there is a general understanding that increasing the levels of all the factors causes a decrease in $\eta_{FTM}$, the measure of performance in the TS FTM nozzle.

As explained in the previous section of 'Interaction Plots', the difference in gradient of the lines in each plot reflects the level of interaction of the various factors in the nozzle. The interactions of NPR against the injection ratio and the area ratio exhibit the most non-parallel relationships out of the all the plots. Out of all the interactions, the lowest level of interaction is reflected between the area ratio and the injection angle simulations.

There appears to be an indication of interaction of the NPR with all of the other factors, as shown by the fact that most of the lines in the ‘vs NPR’ plots appear to be non-parallel. In particular, when the NPR is set at a lower level (the blue lines), most of the other factors seem to achieve significantly better performance when changed from higher to lower levels. With NPR fixed at high levels, the improvement in performance is not as significant. Although the gradient difference of this interaction is not greatly different from the rest of the parameters, it still indicates that keeping NPR at lower levels where possible in the investigation would improve TS FTM performance, and should be adopted. For the application of TS FTM to rocket nozzles, however, keeping NPR at low levels potentially compromises the level of thrust that the rocket is capable of achieving. There is therefore another case for discussion for the design between the level of FTM capability desired and the level of thrust required by the nozzle. It would be useful to the designer if the interactions of other factors could compensate for the restriction of NPR in the design. The next section conducts a detailed study into the understanding of the significance of the factors and their interactions.

4. Pareto plot

A Pareto plot is produced for the factors of the TS FTM cases, as shown in Figure 27. The factors of $A_{ratio}$, $\theta$, $Pr$ and NPR are defined as factors $A$, $B$, $C$ and $D$ in the Pareto plot. The factor $AB$, for example, then represents the factor of the relationship between the $A_{ratio}$ and $\theta$. The blue bar graphs reflect the standardised values of the $E_f$ of the factors divided by the total $E_f$ of the system. The red line with square-shaped data points then reflects the cumulative percentage of the data.

Reading from left to right of the bar chart, the factors are ranked based on their values of $E_f$ relative to the other factors (with the left-most having the highest value of $E_f$). A dotted dividing line is then drawn as the reference line of the system, and is labelled at the most significant $E_f$ change in the bar charts (the initial big ‘dip’ in $E_f$ values). This corresponds to the point where the significant factors are separated from the less significant factors. In this Pareto plot (Figure 27), the reference line occurs at 80% cumulative percentage, where the $E_f$ value changes by 2%, the first greatest decrease across all the factors’ $E_f$ values.

An important observation lies in the fact that the factor $A$, or $A_{ratio}$, is the most significant factor out of the factors in the simulations performed. All three factors that are above the reference line are affected by $A$, which suggests that the changes in $A$ causes the most significant change in the TS FTM.

In contrast, the factor of $C$, or injection pressure ratio, appears to have the least significant effects on the performance of the TS FTM performance. All three of the least significant factors listed in the Pareto plots are affected by the factor of injection pressure ratio. The only more significant parameter that is associated with the injection pressure ratio is that of $AC$ – however this might be attributed to a ‘mixing’ of the effects of the most important factor and the least effective factor on TS FTM performance.

In order to identify the effects of the $A_{ratio}$ on TS FTM, a comparison was conducted between runs 1 and 2, run 1 was where setting $A_{ratio}$ at low levels achieved the best performance, and run 2 was the same simulation performed with $A_{ratio}$ increased. The results calculated from the nozzle parameters read from FLUENT showed that for increases in $A_{ratio}$, the mass flow rate from the primary inlet decreases while the mass flow rate from the secondary inlet increases. The mass flow rate out of the nozzle was however found to decrease with increased $A_{ratio}$, leading to a decrease in momentum thrust. The increase in secondary mass flow rate was attributed to the
dependency of mass flow rate on the cross-sectional area of the geometry. For choked flow in the injector, the velocity and density of the fluid injected is fixed; therefore, the mass flow rate is directly dependent on the last parameter, injector slot cross-sectional area. An increase in the area ratio therefore results in the increase in secondary mass flow rate.

With increased secondary mass flow rate, the jet penetrates further into the sonic flow, hence increasing the induced separation of primary flow from the nozzle wall. This is seen by the higher proportion of blue (low Mach numbers) at the corner of the throat downstream of injection in Figure 28. The increased separation from the nozzle wall directly implies that the nozzle throat is reduced, and therefore the primary mass flow rate of the nozzle is decreased. If acting alone, the decrease in the primary mass flow rate acts to decrease the primary moment thrust, but in the case of the nozzles being compared there is also a bigger region of increased Mach number in the middle of the nozzle for the case with increased area ratio (Figure 28b). These high Mach numbers are caused by the acceleration of the flow after the reduction of the throat, which results in higher velocity at the nozzle exit plane. Since the momentum thrust is obtained through the product of exit velocity and mass flow rate, these two factors discussed act to cancel out each other.

From the results that the author obtained in the calculations, the higher (and hence better) $\eta_{FTM}$ achieved in the best-performing nozzle (run 1) directly implies that the added benefits of thrust reduction in the case with the increased area ratio is more than the cost of the increased secondary mass flow required to achieve it. It would therefore prove more effective to keep the injector area ratio on low levels.

The interaction of the area-ratio and the factor of NPR (factor AD in Figure 27) is also potentially significant in the design of TS FTM. In the literature reviewed (Federspiel 1995), the NPR was identified to not have significant effects on the modulation of thrust. The interaction and Pareto plots, however, have shown that NPR, in particular the effects of NPR on the other parameters, is one of the more significant parameters involved in the process of TS FTM. This could be due to the NPR driving the pressure and therefore the velocity of the primary flow, and since the area-ratio defines the flow rate of the secondary flow based on the primary flow, the setting of the NPR would then affect the amount of mass-flow interaction there is between the flows, and therefore affect the jet interaction between the flows. To conduct an initial test of this theory, the author conducted a trend analysis (see below) for the TS FTM performance based on the secondary to primary flow ratio, to identify if the effects on mass-flow rate proportions would have significant effects on TS FTM performance.
5. Trends analysis

From the interaction plots and factor effects, the author has established that decreasing the levels of the factors results in improvements of TS FTM capabilities. For TS FTM, it appears that changing parameters that result in increased secondary mass-injected, or secondary mass flow rate would decrease the net thrust of the nozzle. The author did a test for this relationship by plotting net thrust ratio (injected over non-injected) cases against injected mass-flow ratio. The plot is shown in Figure 29.

The plot shows that the momentum thrust of the TS FTM nozzle is decreased with increased injection, due to the constriction of the throat effectively reducing the primary mass flow rate. This result also shows that the two effects of adding additional mass-flow into the system through injection and the re-acceleration of the flow after the TS is not significant enough to equal the decrease in momentum thrust caused by the mass flow. Also, the figure shows that pressure drag increases for increased injected mass flow ratio. As the nozzle is over-expanded, the static pressure at the nozzle exit is insufficient to overcome the force exerted on the nozzle by the surrounding atmosphere. Therefore if the expansion of the flow through the nozzle is increased because of the smaller throat (and hence increasing the nozzle diverging area ratio), the static pressure at exit would be lower than non-injection cases, which increases the pressure drag on the aircraft. For designers focused on the magnitude of thrust decrease in their designs, this relationship is the driving force behind further development.

More importantly, however, the results show that the relationship of the mass-flow ratio is one of the driving parameters behind the decrease in thrust. In conjunction with the previous understanding of the interaction effects of the factors in ‘Interaction plots’, the interactions between these factors and the mass-flow ratio should also be studied, and from that study the details behind the interactions and fundamental factor effects can be understood. This study, however, could not be completed within the time of this report, but should nonetheless be investigated by future work.

The ‘scatter’ of the data is a result of the effect of varying different factors for each point. For example, for an injection ratio of approximately 0.2, there are two data points, from run 6 and run 14. The settings for these two runs differ in the fact that run 14 has NPR of 8 whereas run 6 has a NPR of 4. Every other parameter is held constant. The difference in NPR causes a difference in the magnitude of primary and secondary flow rates for the two runs – the primary nozzle flow rate is about 17 kg/s for run 14 whereas the primary flow rate is about 8 kg/s for run 6. This difference in magnitude is amplified by the multiplication of velocity (to find momentum thrust). The pressure thrusts are also different for both runs, as a result of NPR differences, and any differences in this data is compounded by the multiplication of the nozzle exit-plane area. These differences are then divided by the results from another simulation, which increases the effects of the difference in factor levels.

The scatter of points is therefore reasonable for trend analysis, but shouldn’t be taken for its quantitative value due to varied effects of factor level changes. There is also a difference in the computational solutions for the sole case where only first-order convergence is obtained, which may pose as a potential off-trend data point on the plot. In order to remove the ‘scatter’, a non-dimensional analysis of the system needs to be conducted to identify the significant non-dimensional factor groups that affect the FTM effectiveness of the system.

Figure 29. Effect of injected mass-flow ratio on thrust ratio
C. TS FTV Results Discussion

All of the TS FTV simulations were performed with the full mesh-grid nozzle geometry in a 2D plane. The injector was then modelled as a slot, similar to the TS FTM cases. As described previously, the modelling of 2D geometry is appropriate for potential validation in the UNSW@ADFA setup, but 2D models fail to account for possible flow behaviour like the wrapping of the primary flow around the secondary injection, which may also happen in the TS FTV simulations. For the purposes of understanding the trends of varying the factors on the system, however, a design of experiments approach was applied.

The simulation of TS FTV is more complicated than the TS FTM simulations, as the nozzle flow tends to be asymmetric about the centreline, and as such, more complex shock interactions would likely occur. The attempts to deflect the primary flow through the use of the secondary jet may also cause turbulence in both the region immediately downstream to the throat and where the deflected flow is required to turn around one of the corners at the nozzle exit.

The same first-before-second-order simulation setup approach was used for the TS FTV cases, but only half the cases managed to converge with the steady second-order solver. It was observed that all the cases that failed to converge were mesh-grids developed for increased \( A_{int} \) levels. This prompted the review of the mesh-grid for errors in mesh-grid generation or FLUENT case setup problems. At the time of this thesis report no errors were discovered that could have been the source of instability in the TS FTV cases that failed to converge. The unsteady solver was then used to simulate the flow, with the solver settings similar to that used in the Deere nozzle validation study.

Parallel CFD solutions were also run with the co-flow condition in an attempt to stabilise the flow in the nozzle. At the time of this report, the unsteady solutions have yet to show signs of convergence (still exhibit oscillatory behaviour), and as such steady solutions with co-flow at first-order convergence will be used for the analysis that follows.

1. Best-performing TS FTV nozzle analysis

Of all the simulations that were performed, the highest value of \( \eta_{FTV} \) obtained is approximately 1.33 degrees/\%-injection, which is the case of run 5, with an injector-throat area ratio of 0.02, injector angle of 30 degrees, injection ratio of 3 and NPR of 4. The lowest \( \eta_{FTV} \) obtained was that of run 14, with all the settings of the previous setup but an injector-throat area ratio of 0.04 and NPR of 8. The Mach number distribution in the flow field is shown in Figure 30.

The flow-field appears to exhibit the same oblique shock behaviour that has been observed previously in this thesis. The difference in this case is the fact that there is only one injector instead of the dual or reflected injectors in the TS FTM case. The injector slot is therefore the only source of disturbance to the flow.

As such, the compression wave that is generated by the interaction of the primary flow turned by the secondary injection into the nozzle walls does not have a ‘counterpart’ on the opposite side of the nozzle. This is the reason why the initial sonic lines (the initial green zones) appear to be skewed to the side of the injector in the nozzle flow field, and is the primary cause of the primary nozzle thrust deflection.

The compression wave that is generated is free to move across the cross section of the flow field, causing decreases in the flow velocity across itself. This process turns the flow further and aligns itself along the lower face of the nozzle. The best performance case has a flow field with a strong compression wave spreading across the nozzle diverging section, and also an oblique shock interaction at the exit plane of the nozzle that is

![Figure 30. Mach number distribution of TS FTV nozzle with best performance](image-url)
characteristic of over-expanded nozzles. The optimum TS FTV conditions based on the effectiveness factor defined could possibly be achieved by operating at the nozzle conditions such that the compression wave generated just impinges on the opposing face of the nozzle.

The turning effect of the compression wave theoretically exhibits similar relationships as that of the SVC FTV method. The difference in the compression wave and the oblique shock generated in the SVC method is that the compression wave occurs over a longer length along the nozzle, and therefore causes a lower entropy generation, or loss of total pressure when turning the flow.

It should also be noted that the compression wave is not the main factor of thrust deflection, but rather a secondary effect caused by the shifting of the throat. The compression wave should therefore be used as more of a trend gauge, with the ‘just impinging’ case being investigated to identify whether the existence of the compression wave generates similar turning effects as the oblique shock generated in the SVC method.

The degree of thrust deflection in the TS FTV simulation results is lower compared to other data obtained from experiments conducted in other literature (see Appendix A). The best performing (in terms of effectiveness) case only achieved a thrust deflection of 3.67 degrees. Referring back to the background summary table in Appendix A, TS FTV is known to be capable of achieving up to 20 degrees of primary flow deflection in a dual throat FTV nozzle. There is therefore an understanding that geometry modification beyond that of the basic converging-diverging nozzle used in this particular study may be required in order to achieve higher thrust vectoring angles. For example, the addition of a second throat (Deere 2003) is expected to increase the region of separation at the downstream region of the throat – this increases the pressure gradient between the primary flow and the area of separation, which turns the flow and shifts the sonic line further.

The highest thrust-vectoring angle achieved in this thesis was 5.9 degrees in run 8 – a nozzle geometry with the biggest injector-slot size, 50 degrees angle of injection, and injection pressure ratio of 3 at a NPR of 4. In order to achieve this flow, however, 11% of the primary flow was required in the injected flow. This is not feasible on an actual aircraft, with problems obtaining the additional pressure to supply 3 times the primary nozzle pressure into the injector.

2. Interaction plots

The interaction plots in Figure 31 are those of TS FTV, and were produced using the same process as described in the TS FTM interaction plots. For the purposes of analysis, $\theta_{\text{ratio}}$, $\theta$, $Pr$ and $NPR$ are defined as factors $A$, $B$, $C$ and $D$ respectively.
Most of the lines plotted have a shallow gradient, which suggests that the value of $\eta_{FTV}$ is mostly independent of the factor levels being varied in the x-axis.

The factors also appear to interact weakly with each other, with most of the lines only having slight changes in gradient for factor changes. The only significant interaction from the plot is the significant change in gradient for the AC and CD plots. These suggest that there is an antagonistic relationship between the factors involved within the range defined; in other words, changing the levels of one of the factors, for example C in the CD plot, would result in the opposite effect with varying the levels of the other parameter, in this case D.

In particular, for the interaction plots concerning factor A, changing the level of factor A appears to alter the gradient of the lines, especially in the case of the AC plot, where changing the level of A changes the gradient of the line from a gentle positive gradient to a steep negative gradient.

The Pareto plot is again produced using the same process as the TS FTM case, with the factors being labelled as A, B, C and D as explained before. The Pareto plot for TS FTV case, however, is very different from the TS FTM Pareto plot.

From the comparison of the height of the bar charts, A is the most significant factor in the process. In this case, however, the factor takes up 25% of the cumulative achieved performance, almost twice as much as the next most important individual factor, C. In contrast to TS FTM where C was the least significant factor, the factor B is the least significant individual factor in the Pareto plot. On average, however, the least significant factors are those with relationships with the factor D.

The area ratio and its interactions with the injection pressure ratio and injector angle have the most significant effects on the development of the nozzle. Changing the parameter of area ratio has the effect of varying the mass flow rate in the injector, which in the TS FTV context is likely to cause increased flow separation at the throat. This increased separation induces more turning of the primary flow back towards the nozzle wall, therefore generating a stronger compression wave upon contact with the nozzle face, which further vectors the primary flow.

The parameters of the area-ratio and the injector angle of the nozzle (factors A and C) have often been studied in previous literature (Deere 2003) - the investigations conducted now shows that there is an additional relationship between the two factors that needs to be addressed in future investigations. In other words, there might be a fundamental interaction between the factors that have not been taken into account in previous one-factor analysis. Varying of the area-ratio might affect the ratio of the secondary-primary flow rates, however the angle at which the secondary flow is injected into the primary flow decides the amount of relative momentum the secondary jet has with respect to the primary flow. The amount of jet penetration is therefore very likely a result of the interaction and combination of these two factor levels.

Judging from the fact the injector angle was not a significant factor affecting TS FTM, but yet a significant parameter affecting TS FTV, it can be reasoned that the injector angle has an effect on the ‘skewness’ generated by the flow, which could be reflected by the angle of the compression wave to the wall. The effect of ‘skewness’ would not be as great in TS FTM due to the symmetry nature of the flow. Therefore, the interactions of the area-ratio and the injector angle could have an interacting effect as the injector angle would determine the amount of shear-interaction between the secondary flow rates determined by the area ratio, and hence varying the amount of skewing of the sonic plane and therefore the net thrust output angle.
4. Trends analysis

In order to facilitate the understanding of the important parameters, a plot showing the effects of the injected mass-flow ratio on the degree of thrust deflection is shown in Figure 33.

From the graph, most of the cases exhibit a linear relationship between injected mass-flow ratio and the net thrust deflection. This is in line with the theory that was discussed in the analysis of the best-performing TS FTV nozzle. For a designer that requires the highest amount of TS FTV achievable in a conventional converging-diverging nozzle, the designer has to modify the parameters such that the mass-flow ratio of injected to primary flow is as high as possible. It is worth noting, however, that the effectiveness parameter decided in this thesis does not account for the loss of thrust in the flow in achieving the FTV – as such it is possible that the thrust is deflected to the largest degree in the high injection-mass-flow ratio case, but the thrust may be reduced significantly in the process, which compromises aircraft performance. In order to address this problem, the effectiveness parameter should be re-defined in future applications to perhaps involve the turning momentum of the flow in the measure of effectiveness.

Generally, \( \eta_{FTV} \) (as defined by this thesis) is higher in the mass-flow ratio range of 2-3%, with the highest effectiveness achieved by using a mass-flow ratio of 2.8% achieving a \( \eta \)-value of 1.4, as shown in Figure 34. This is in line with most of the background literature reviewed (see Appendix A) – in particular, most of the NASA Langley experiments on TS FTV experiments were conducted with a mass-flow ratio of 3%. It is also observed that in general, increasing the injected mass-flow ratio causes a decrease in effectiveness of \( \eta_{FTV} \) – this suggests that the gain of increased secondary mass-flow penetration turning the flow does not achieve proportional returns in FTV.

The results therefore suggest that the combination of factors used should aim to reproduce a mass-flow ratio in the optimum range of 2-4%, and any combination of factors that exceed this range of relationships may
encounter a decrease in performance (although the net thrust vectoring angle increases). Again, a study should be conducted in the varying of the identified factors to achieve this mass-flow ratio.

The source of the ‘scatter’ is similar to that described previously in the TS FTM section, a combination of the effects of varying different factors for each data point on the plot, and the lack of complete iterative convergence (especially for the unsteady solutions) at the time of this thesis report. Again, a non-dimensional analysis needs to be conducted to identify the factor groups that affect the effectiveness of TS FTV experiments.

VIII. Lessons Learnt

From the CFD-simulations the author has learnt about the use of three software packages – CATIA, GAMBIT and FLUENT. The author learnt through the process of trial and error and post-trial analysis that the CFD-simulation process is highly complex, and heavily dependent on the spatial and temporal dimensionality of the mesh. The author also learnt that the value behind CFD lies in its capability to predict the trends and behaviour of a flow without the physical set-up of experiment, and the potential to look at the results generated and highlight details in the flow that would otherwise be difficult to capture.

Through the use of the Design of Experiments approach, the author has learnt about the approach to the evaluation of a design and the discussion of important parameters in the flow. The author has learnt to adapt the tools available to the understanding of the details of the flow, and also to identify the significant parameters that affect the behaviour of the flow. The author also learnt that a single factor experiment has its limits, especially in the (lack of) prediction of the effects on different factors on each other. In order to identify the effects and conduct a full design investigation, a process such as the one adopted by the author needs to be conducted.

IX. Conclusions

The aim of the thesis was to identify the significant parameters in the processes of TS FTV and TS FTM. After conducting a complete full factorial design of experiments, key factors were identified that affected the performance of the two processes under study. For both cases, the non-dimensionalised value of injector-slot size to throat size was identified as an important parameter for the studies. On top of this result, however, significant factor interactions were also identified.

In the TS FTM case in particular, the interaction plots and Pareto plot revealed that the effect of the injector-slot size to throat size and the interactions of the parameter with the NPR and the injector angle had significant effects on the performance of the nozzle, and therefore should be looked at in further detail. Low levels of the factors of the NPR, injector angle and the minimal area-ratio factors should be applied in TS FTM applications.

In the case of the TS FTV process, the interactions of the area-ratio parameter with the injection ratio and injection angles were also found to be significant. The area-ratio and injection angles as described above should be kept to as small a value as possible with the injection ratio should be maintained at a high level to achieve optimum TS FTV cases. The relationships between these factors identified should also be closely monitored in the identification of best TS FTV designs. In the case of TS FTV especially, the mass-flow ratio should be kept to a value between 2-4%, as was both discovered by the literature and from the simulations.

Trend analysis as a method for further understanding the flow also showed that in general increased secondary to primary mass flow ratio increased the thrust modulation or thrust vectoring that the TS method could achieve. The mass-flow ratio was identified by the author as a significant output parameter from the varying of the factors, and as such the optimum mass-flow ratio needs to be identified based on the pressure settings and nozzle geometry enhancements proposed.

X. Recommendations

For future work, the author would recommend the completion of the TS FTV cases using the unsteady solver. Similarly, the TS FTM cases can be modified to perform axisymmetric studies of the TS FTM, as discussed in this report. A full mesh dependence study across all meshes generated is also advised in the future pursuit of design of CFD experiments. A full study of the identified factors and their interaction with each other also needs to be conducted, with the mass-flow ratio possibly being an additional parameter that needs to be reviewed.

The design of experiments approach could also be re-done using the discovered ‘optimal’ nozzle design as the central level, and using a 3-level factor approach to evaluate in additional detail the effects of different factors on the effectiveness of TS FTM and FTV.

In terms of validation of the simulations, there is much potential in the development of the UNSW@ADFA test rig, and also the production of both 3D computational cases and setup in attempts to more accurately model the TS method in both applications. The results should also be compared with a simple analytical solution of the flow in order to achieve better validation.
Otherwise, there is also room for the development into 2D extension of the converging-diverging nozzle by introducing recessed cavities and additional injectors. Possible further enhancements include the addition of more injectors at different locations (for example, inclusion of injectors at the nozzle flaps to obtain the added benefits of SVC in FTV).

There is also additional scope for the investigation of the parameters used for determining TS FTV and TS FTM effectiveness – a non-dimensional analysis may uncover non-dimensional groups which help to explain the effects of the factors on jet penetration. There is also a possibility of comparison between the SVC FTV efficiency factor and the TS FTV effectiveness factor.

Acknowledgements

The author would like to acknowledge Dr Andrew Neely and Dr John Young for the keen supervision over the process of this thesis, and for simply being very approachable wealths of knowledge that helped the author understand more about the complicated thesis process. The author would also like to acknowledge Dr Sean O’Byrne for willingly sharing valuable insights into the potential development of the thesis. Finally, the author would like to acknowledge PLTOFF Brendan Blake for voluntarily sharing his research and CFD knowledge with the author.

Appendices

A. Background Research Summary Table
B. Client Brief
C. Detailed Task Lists
D. Milestone chart
E. Summary Table of Results
F. Mach-number Distributions of Results

References


