COMPUTATIONAL INVESTIGATION OF FLUIDIC COUNTER FLOW THRUST VECTORING APPLIED TO SMALL ENGINE EXHAUST JET

Farzad Banazadeh  
MSc Student, Department of Mechanical Engineering, The University of Hormozgan  
fa.banazadeh@gmail.com

Saeid Niazi  
Assistant Professor, Department of Mechanical Engineering, The University of Hormozgan  
saeidniazi@hotmail.com

Afshin Banazadeh  
Assistant Professor, Department of Aerospace Engineering, Sharif University of Technology  
Banazadeh@sharif.edu

Abstract
A computational investigation of fluidic counterflow thrust vectoring on a circular-shape nozzle, integrated with a Coanda surface has been conducted. Three-dimensional numerical simulations were run using the computational fluid dynamics code with two-equation standard $k$-$\varepsilon$ turbulence model. In the counterflow concept, thrust vectoring is achieved by applying a vacuum to a slot adjacent to a primary jet that is shrouded by a suction collar. Counterflow thrust vectoring is an innovative technique that uses no movable components to redirect a thrust producing jet. This geometry relies on a curved surface (named collar), and a secondary section that was divided into four identical sections for the purpose of multidirectional vectoring and cooling. The purpose of the present study is to investigate the thrust deflection angle for different primary flow and gap sizes between collar and nozzle with the use of different vacuum pressure. For validation, computational results were compared to experimental and computational data obtained at Cranfield University with co-flow method for this geometry. In general, computational results were in good agreement with experimental performance data, indicating that efficient thrust vectoring can be obtained with low secondary flow requirements compared to co-flow method.

Key words: Thrust Vectoring, Counter-Flow, Fluidic, Coanda surface, Computational Fluid Dynamics

Nomenclature
\begin{align*}
\alpha & : \text{Measured Thrust-vector Angle} \\
R & : \text{Coanda Surface Radius} \\
c & : \text{Coanda Surface Height} \\
p_{s} & : \text{Pressure At Suction Slot Exit} \\
\dot{m} & : \text{Air Mass-flow Rate} \\
T & : \text{Measured Thrust Value} \\
G & : \text{Measured Secondary Slot Height} \\
x, y, z & : \text{Cartesian Coordinates} \\
TV & : \text{Thrust Vectoring} \\
FTV & : \text{Fluidic Thrust Vectoring} \\
CFTV & : \text{Counter Flow Thrust Vectoring} \\
CFD & : \text{Computational Fluid Dynamics}
\end{align*}
1. Introduction

Thrust vectoring is the ability of an aircraft or other vehicle to deflect the angle of its thrust away from the vehicles longitudinal axis. Up to the present time, Studies have shown that thrust vectoring (TV) concepts can provide many benefits to the modern fighter aircraft. Thrust vectoring research was initiated more than thirty years ago. The maneuverability of aircraft is traditionally achieved by the use of aerodynamic control surfaces such as ailerons, rudders, elevators and canards. The deflection of these surfaces creates changes in the aerodynamic forces and moments acting on the vehicle and causing it to maneuver. An alternative to such classical methods is thrust vector control which has successfully been identified and demonstrated many potential benefits such as enhanced aircraft maneuverability, survivability to high-performance aircraft. The ability to land and take off from short unimproved runways, agility and reduction in size or complete removal of air vehicle control surfaces is also improved with TV. Moreover, The ability to perform transient maneuvers at low speeds and high angles of attack is a foremost benefit of thrust vectoring.

There are two methods of achieving thrust vectoring either conventional methods, which rely on mechanical means and the most recent methods, which are fluidic-based thrust vectoring techniques. First studies were based on mechanical techniques in which movable vanes, nozzles or plates vectored the jet. A promising area of research is fluidic nozzle control. The fluidic thrust vectoring (FTV) is based on using a secondary air stream to influence the behavior of the primary jet. Fluidic control requires few or no moving parts in the primary nozzle and several studies have demonstrated the successful use of fluidics compared to conventional mechanical thrust vectoring systems to control nozzle flow. However, increase of weight, system complexity, usage of special materials, need continuous maintenance, difficult to integrate, aerodynamically inefficient and low response time, typical of mechanical systems, constraint aircraft performance. While, fluidic-based methods offer reduced weight, higher reliability, faster dynamic response and stealth compatibility.

Up to the present time, a variety of different methods of FTV have been developed, which include the use of counterflow and co-flow fluidic thrust vectoring, throat shifting, shock vector control (SVC), combined methods. Deer presented several experiments to compare different thrust vectoring techniques. The results show that throat shifting concept produces better nozzle discharge coefficients while the SVC method results in higher vector angles. Counter-flow and co-flow are different from the others since they use external secondary flow traveling in opposite or in direction of the primary nozzle jet, respectively. The fluidic method used in the current study uses a secondary counterflowing stream to vector the primary jet. The counterflow thrust vectoring concept was first reported by Strykowski and Krothapali.

The work presented in this paper deals with the development of a (CFTV) system by means of a real micro-jet engine relies on a phenomenon known as Coanda effect. In the counterflow concept, thrust vectoring is achieved by applying a vacuum to a slot adjacent to a primary jet which is shrouded by a suction collar. The counterflow thrust vectoring concept is shown in Fig.1. The vacuum creates a secondary reverse flowing stream near the primary jet. The shear layers between the two counterflowing streams mix and entrain mass from the surrounding fluid. Coanda effect is the natural tendency of fluid and gaseous jets to attach to the wall, which is projected close to them and follow the convex curvature of the solid boundary. In this phenomenon air is exhausted out of the peripheral jet nozzle. It then passes over a convex surface (collar). Skin friction occurs between the fluid and the surface, which tends to slow the moving fluid. This resistance to the flow of the fluid pulls the fluid, both secondary and main, towards the surface, causing it to adhere to a solid surface forming the
Coanda effect. Decreasing the jet velocity causes increase surface pressure so that it equals the sub ambient pressure. So the jet flow separates and a new stagnation point and stagnation streamline are formed on the trailing edge of the Coanda surface.\textsuperscript{5,6}

Parameters that affect the Primary flow to redirect it include, main stream velocity, Secondary vacuum pressure, slot size and radius of curvature of collar. In the present work, configuration is a cylindrical duct with a bell mouth exhaust, divided into four identical sections for multi-directional thrust vectoring as shown in Fig. 2. In this way, by placing the circular shape collar surface at the exit of the engine exhaust nozzle and applying suction along the upper slot is that the only active slot and the secondary counter-flowing stream was established parallel to the Coanda surface then CFTV can be obtained.

![Figure 1: Counterflow Thrust Vectoring Concept](image1.png)  
![Figure 2: Configuration of Fluidic Nozzle](image2.png)

In a conceptual understanding of CFTV method, vectoring achieved with the generation of a “countercurrent” shear layer along the suction side of the jet. As shown in Fig. 3 the countercurrent shear layer has a fundamentally different character than a traditional “coflowing” shear layer. Hence, there is an increase in the level of vortical and turbulent activity in the shear layer as it becomes globally selfexcited. Because of these results, the mixing is enhanced and consequently, a countercurrent shear layer can have a growth rate more than 50\% higher than a coflowing shear layer. Due to increase in vortical and turbulent activity in the countercurrent shear layer, it entrains more mass from the surrounding ambient fluid than the coflowing shear layer. This, in turn, creates asymmetric pressure gradients about the nozzle centerline and draws the jet off-axis. Essentially, continuous control of the vector angle can be obtained because the instability level of the countercurrent shear layer depends on the amount of suction applied.\textsuperscript{1}

![Figure 3: Countercurrent and Coflowing Shear Layers](image3.png)

This method was used in this investigation to vector the exhaust gases of the AMT Olympus micro-jet engine as a propulsion system in order to computational analysis.\textsuperscript{12} The goal of this study was to investigate with a computational simulation a relationship between the vectored thrust angle and key parameters such as secondary slot height, primary mass flow inlet and secondary vacuum pressure. For validation, computational results were compared to experimental and computational data obtained at Cranfield University with co-flow method for this geometry.
2. Computational Fluid Dynamics Simulation

Prediction of counter flow thrust deflection angles relies on the accurate numerical simulation of shear flows. Moreover, Such flows are very sensitive to turbulence modeling. For this reason, in this investigation an extensive turbulence model validation was performed to ensure that our turbulence modeling, numerical grids and simulations agree with the best results. Computational fluid dynamics simulation is utilized to predict the response of the system. When a reliable CFD model simulation is extended, it will be used for geometric optimization by saving time without the need for extensive and expensive experimental equipment.

The scripting language and journal file method with the ability to define and use variables in some CFD meshing software produce a scripted record of the user interactive session and also provide parametric study capability. Thus, This feature allows us to safely examine the effect of one or more geometric parameters on the model performance, when designing a FTV system. CFD offers unique advantages compared with experimental tests. For example in this investigation it can check easily the effects of secondary air mass flow rates and different gap sizes that are beyond the maximum value that achieved in the experimental tests without the need for expensive experimental facility. Moreover, It can also compute the thrust-vector angle without the test errors such as air-leakage.

2.1. Flow Solver and Governing Equations

For the present study, the numerical solution was computed using a 3-D finite volume code to assess the performance of FTV system. This code with 2-equation turbulence modeling was chosen for the current simulations because it was developed for and has been extensively used to accurately predict propulsive flows with mixing, separated flow regions, and jet shear layers. In this code, a centered finite volume technique was used to solve the Reynolds Average Navier-Stokes (RANS) equations assuming conservation of mass (continuity), momentum and energy for a perfect gas.

Generally, in choice of numerical method and turbulence models an early prediction of a general fluid flow can be useful. The 2-equation k-ε turbulence model, with second order closure, is used to model more complex viscous flow features and This standard k-ε model was selected to investigate the effects of turbulence in flow. k-ε turbulence model is one of the most prestigious models. This is the simplest turbulence model for which only initial and/or boundary conditions need to be supplied. It owns excellent performance for many industrially relevant flows. Based on previous experience, the segregated solver, with SIMPLE pressure-velocity coupling, is utilized. Standard pressure and second-order upwinding is used to discretize the convective terms in momentum equations with second order central differencing used on the viscous terms. Second-order upwinding scheme is also used for energy equation. This strategy speeds convergence and reduces computational time. As mentioned above equations such as conservation of mass, momentum and energy and also two-equation k-ε turbulence model were used in this paper. Equations of k-ε are as follows:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho ku_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon - Y_M \tag{1}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \mu + \frac{\mu_t}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} G_k \frac{\varepsilon}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \tag{2}
\]

\[
G_k = -\rho \mu_t \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \tag{3}
\]

\[
Y_M = 2\rho \varepsilon \frac{k}{\gamma RT} \tag{4}
\]
\[ C_\text{f} = 1.44, \quad C_\mu = 1.92, \quad C_\sigma = 0.09, \quad \sigma_\text{f} = 1.0, \quad \sigma_\mu = 1.3 \]  

(5)

### 2.2. CFTV Numerical Simulations

In this paper different geometric and fluid design variables will be investigated to define the jet deflection associated with CFTV systems. Table 1 shows the values of the Computational variables and the geometric parameters. To compute the deflection angle from the numerical results, the surface integration of velocity vectors or momentum on the adjoining plane precisely upon the nozzle in x and z directions are used to estimate the jet deflection angle, \( \alpha \)

\[
\alpha = \tan^{-1}\left( \frac{T_x}{T_z} \right)
\]

Figure 5 shows the contours of total temperature with no vacuum pressure in secondary slots and no thrust vector angle (0\(^o\)).

![Figure 5: Contours of Total Temperature with 0% of Flow Ratio and Thrust Vector Angle 0\(^o\)](image)

#### Table 1: Computational Flow and Geometric Properties

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Notation</th>
<th>Comment</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary mass flow inlet</td>
<td>( \dot{m}_p ) (kg/sec)</td>
<td>Variable</td>
</tr>
<tr>
<td>Secondary vacuum pressure</td>
<td>( P_\text{s} ) (kpa)</td>
<td>Variable</td>
</tr>
<tr>
<td>Secondary slot size</td>
<td>( G ) (mm)</td>
<td>Variable</td>
</tr>
<tr>
<td>Coanda surface radius</td>
<td>( R ) (mm)</td>
<td>Fixed</td>
</tr>
<tr>
<td>Coanda surface height</td>
<td>( c ) (mm)</td>
<td>Fixed</td>
</tr>
</tbody>
</table>

### 2.3. Nozzle Geometry

The current nozzle geometry is intended to enhance the thrust vectoring capability of CFTV method by manipulating flow separation and mixing shear layer in a collar region. The collar section was located after the cylindrical duct. At location where the duct and collar are connected, there was a secondary section that divided into four identical sections for the purpose of multidirectional vectoring and cooling. In order to find the best thrust deflection angle numerous simulations were computed to examine the effect of various nozzle design parameters.
2.4. Computational Grid Domain

As shown in Fig.4 the computational grid for the CFD model extended from the engine exhaust and secondary exit to the far-stream conditions in the horizontal axis, where the control volume exposes the collar surface and the exhaust walls. In this CFD model the computational mesh was defined with 7 blocks in a three-dimensional domain to define the internal passages and the free stream volume. Moreover, it is intended that all the blocks are composed of structured grids. The entire grid system consists of 849,800 cells. The grid points were set to produce fine mesh close to the walls for the boundary layer, to remain $y+$ parameter in the limit of the law-of-wall ($30 < y+ < 500$). Hence, no wall treatment or adaptation was utilized in the solver.

2.5. Boundary Conditions

The upstream boundary for primary nozzle was set to mass flow inlet condition, where the primary mass flow rate was calculated from the characteristics of the AMT Olympus micro-jet engine as a propulsion system. Detailed specifications of the engine are listed in table 2. The upstream boundary for secondary nozzle was set to pressure outlet condition, so that this vacuum pressure was adjustable about less than 1% of the primary mass flow rate for each active part based on an optional vacuum pump. The primary nozzle flow and the fluidic suction flow were specified with a fixed total-temperature boundary condition. The downstream boundaries were set to pressure outlet condition with zero gage pressure. The air was also assumed to be governed by the ideal gas law. A no-slip, adiabatic wall boundary condition was implemented on nozzle surfaces to obtain viscous solutions.

<table>
<thead>
<tr>
<th>Table 2: The AMT Olympus Engine Specifications</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thrust (max. RPM 112,000)</td>
</tr>
<tr>
<td>Pressure ratio</td>
</tr>
<tr>
<td>Mass flow</td>
</tr>
<tr>
<td>Fuel flow</td>
</tr>
<tr>
<td>Dimensions</td>
</tr>
<tr>
<td>Maximum exhaust temperature</td>
</tr>
</tbody>
</table>
2.7. Results and Discussion

Figure 6 shows the contours of total temperature for three cases with constant vacuum pressure which expresses the relationship between thrust-vector angle and different slot height at constant primary mass flow inlet. As shown in this Fig, with reducing the secondary slot height from 2mm, thrust-vector angle would increase from 2.5 degree in a constant vacuum pressure at 42 kpa and primary mass flow inlet at 0.31 kg/s. Figure 7 shows the relationship between thrust-vector angle and different vacuum pressure for three slot heights. As shown in this Fig, increasing the vacuum pressure causes increase of thrust-vectoring angle. In other words, increasing flow ratio improves the thrust vector angle, where the vector angle increases rapidly with increasing vacuum ratio. Also, another result that is driven from this figure is that, increasing the secondary slot height more than 1 mm results in reduction of thrust-vectoring angle. This could be due to momentum reduction, which is a principal parameter in FTV concept.

Figure 8 shows the thrust-vectoring angle achieved for 1.4 mm slot height and 0.21 kg/s primary mass flow rate in co-flow method. These results were obtained by computational (CFD) and experimental (thermal camera and load cells) methods. As seen in this figure the difference between thermal camera and load cell and CFD results were mainly because of the thermal expansion of struts and nozzle that made it floppy during the tests especially in high shaft speeds and difficult to recognize in thermal photos. More details are given in ref. 4. Moreover, according to this results for obtain the deflection angle about 8 degree in this way for co-flow method approximately more than 6 percent of secondary to primary air mass flow rate was needed. While, as seen in Fig. 9 the thrust-vectoring angle achieved for 1.4 mm slot height and 0.21 kg/s primary mass flow rate for counterflow method that was needed about 3 percent of secondary to primary air mass flow rate to obtain the deflection angle about 8 degree.

This results means that with a smaller percentage of secondary to primary air mass flow rate in the counterflow method can be gain the same angles of deflection in co-flow method. This method causes largest deflection obtained with the least expenditure of energy and also, requires equipment and facilities necessary to create secondary flow becomes smaller and lighter. Regarding to the results, it is seen that the experiment and computation show a similar trend, with regimes of coflow and counterflow. Important features to note are, first, that both the experiment and computational simulation demonstrate that vectoring can be attained with coflowing and counterflowing secondary streams. In addition, both the experiment and the computation had very low secondary air flow rates, approximately 0.3-7.5% of the primary jet flow rate, for α values ranging from about 2 to 9 degree. This result confirms that counterflow thrust vectoring has minimal secondary flow requirements. This indicates that counterflow thrust vectoring is relatively efficient.

With comparing the two Figures 7 and 9, results shows that with increasing the primary air mass flow rate from 0.21 to 0.31 kg/sec for two identical geometry and flow properties the vectored angle was reduced. For the fluidic nozzle to be practical for use on an aircraft, the structure of geometry, equipment and facilities must be kept as small as possible to reduce base drag and weight on the aircraft. Therefore, This indicates that counterflow thrust vectoring is relatively efficient.
Figure 6: Contours of Total Temperature (k)

(T.V. angle achieved in $\dot{m}_p=0.31$ kg/s and $p_s=42$ kpa for three slot heights)

Figure 7: T.V. Angle against different Secondary Vacuum Pressure ($\dot{m}_p = 0.31$ kg/sec)

Figure 8: Thrust-vector Angle against Secondary Air Mass Flow Rate with Co-flow Method

( $\dot{m}_p = 0.21$ kg/sec)
3. Conclusions

- A computational study of fluidic counterflow thrust vectoring has been conducted. Three-dimensional numerical simulations were run using the computational fluid dynamics code with two-equation standard k-ε turbulence model.
- In this paper the implementation of fluidic control to vector thrust of a small gas turbine was presented and counterflow technique was examined and found to perform well for low gas turbine shaft speeds.
- Based on computational results, a coflowing and counterflowing secondary stream revealed new details about the generation of a countercurrent shear layer and its relation to secondary suction. In both technique, Although it was limited in spread and existed only within the suction collar, but a countercurrent shear layer was generated. In the coanda surface, Since the reverse velocity ratios were large enough It can cause local absolute instability, but the jet itself did not achieve levels of global absolute instability necessary for self excitation. This agrees with results reported by Strykowski and Krothapalli et. Al.
- The computational results show that increasing the slot height more than its designed value (1 mm), causes a nonlinear reduction of thrust-vectoring angle. Also, in a fixed geometry and secondary vacuum pressure, with increasing the primary mass flow rate causes a notable reduction of thrust-vectoring angle. On the other hand, this numerical results show that increasing the vacuum pressure causes increase of thrust-vectoring angle.
- The results of the study indicate that CFTV is sensitive to the geometric and flow variables such as suction/blowing slot height, and suction/blowing intensity. Also, with a smaller percentage of secondary to primary air mass flow rate in the CFTV method can be obtain the same vectored angles in co-flow method.
- In future, a more detailed sensitivity study for geometric and flow variables will be attempted. A related study is in process to investigate the efficiency of such a system for different collars and flow properties.

Figure 9: Thrust-vector Angle against Secondary Air Mass Flow Rate with Counter-flow Method

\[(\bar{m}_p = 0.21 \text{ kg/sec})\]
References


