UNIVERSITI TEKNOLOGI MARA

University Publication Centre (UPENA)

Journal of Mechanical Engineering

An International Journal

Volume 6 No. 2 December 2009 ISSN 1823-5514

* Technical Note

JOURNAL OF MECHANICAL ENGINEERING (JMechE)

Journal of Mechanical Engineering (ISSN 1823-5514) is jointly published by the Faculty of Mechanical Engineering (FKM) and Pusat Penerbitan Universiti (UPENA), Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia.

The views, opinions and technical recommendations expressed herein are those of individual researchers and authors and do not necessarily reflect the views of the Faculty or the University.

Journal of Mechanical Engineering

An International Journal

Volume 6 No. 2 **December 2009** **ISSN 1823-5514**

* Technical Note

Interference Effects between Duct and Control Vane on a Micro Air Vehicle

Sheila Tobing
Tiauw Hiong Go
Roxana Vasilescu
School of Mechanical & Aerospace Engineering
Nanyang Technological University
Singapore 639798, Republic of Singapore

ABSTRACT

A numerical study on the interference effects between duct and control vanes on a Vertical Takeoff and Landing Micro Air Vehicle is presented in this paper. The numerical analysis is conducted systematically with the solver first validated before it is applied to the full vehicle simulations. Different combinations of duct and control vanes are tested to determine the interactions between the two components and their effects on the vehicle's aerodynamics. The simulations show that the duct influences the local flow around the control vanes. The duct redirects the flow before it passes the control vanes, altering the aerodynamic forces and moment generated by the vanes as well as the vehicle. The control vanes generate lower aerodynamic forces and moments under the influences of the duct, compared to when they are simulated as isolated (individual) components. The over-predictions of the vehicle's aerodynamic forces and moments need to be taken into consideration in designing as they will introduce a high error to the design.

Keywords: *micro air vehicle, component buildup, interference effect, duct, control vane*

Introduction

The use of ducted-propeller and control vanes is quite common in a Vertical Takeoff Landing Micro Air Vehicle (VTOL MAV). Adding a duct or a shroud to a

ISSN 1823-5514

propeller offers advantages such as higher static thrust, better noise suppression and better protection from foreign object damage (FOD) [1, 2]. A control vane is typically used to gain control in the longitudinal, lateral and directional modes as well as to counteract the torque generated by the propeller [3, 4]. The control vane deflection generates forces and moments that affect the aerodynamic characteristics of the vehicle. The duct and the control vane, individually and independently as a component, may improve the performance and/or the stability of the vehicle, but little is known about the interactions between them. In predicting the aerodynamic coefficients of ducted fan MAVs, a program named AVID OAV takes into consideration the influence of duct on inside-the-duct control vanes by including the velocity induced by the ducted-fan in calculating the vanes' lift and drag [5, 6]. In another work, a generic modular-simulation-model of ducted-fan vehicles is built by including the effect of duct on the velocity, angle-of-attack and sideslip angle of a vane [7]. These works focus on formulating empirical equations to predict the aerodynamic forces and moments of ducted-fan MAVs. Unlike previous researches, the current work uses computational data to predict the aerodynamic coefficients and gives emphasis to flow phenomena around the vehicle. Moreover, this research is specifically conducted to analyze the interference effects between the duct and the control vane on a VTOL MAV. The duct and various configurations of the control vane are investigated numerically using a RANS solver (FLUENT) to obtain the aerodynamic data and to visualize the flow around the vehicle.

Numerical Analysis Validations

Validation is a critical process of any numerical investigation. The complex flow around MAVs emphasizes the need of this process. Prior to the analysis of the full VTOL MAV configuration, the solver need to be validated numerically. The solver (FLUENT) is tested to examine its accuracy in predicting the complex flow around a low aspect ratio lifting surface, which is typical for MAVs. The case selected for the solver validation is a low aspect ratio annular airfoil/duct with Clark Y airfoil section, the experimental data of which are available. The duct was one of the five models that was investigated experimentally by Fletcher [8].

The duct studied in [8] has a Clark Y airfoil-section and an aspect ratio of 1, which means it has an equal length of diameter (*d*) and chord (*c*) of 16 in or 0.4064 m (Figure 1). The airfoil section is Clark Y with a maximum thickness ratio of 0.117. The area of the duct is 0.165 m^2, calculated using the following equation $A = d \times c$. The duct is tested at a dynamic pressure of 1192.2 Pa, which corresponds to a Mach number of 0.13 and a freestream velocity of 44 m/s. The computation is conducted using Spalart-Allmaras turbulence model, SIMPLE scheme and second order upwind discretization.

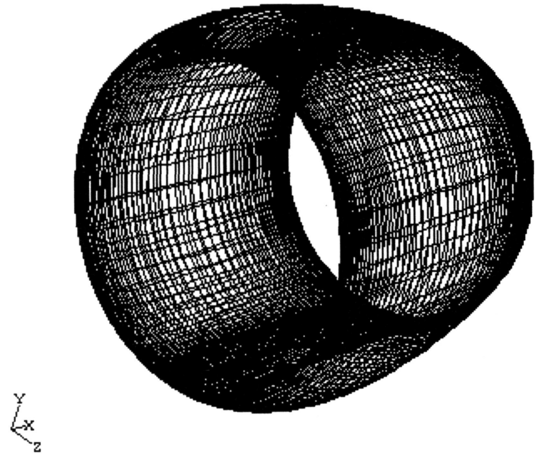

Figure 1: Computational Model of Fletcher's Annular Airfoil/Duct

In the computation, two methods are used to impose the boundary condition to simulate angles-of-attack and sideslip-angles in the computation. The first method uses the vector components of the velocity to represent the angle. The velocity is set as its x, y or z components with respects to the angle simulated. The second method to impose the angle of attack or sideslip angle is by rotating the object such that it creates an angle with respect to a certain axis (Figure 2).

The unstructured grid model is built to accommodate the object rotation method, while the velocity vector method is simulated using the structured grid model (Figure 3). In the next steps, the unstructured and structured grid model will be called as fixed duct and rotated duct, respectively. The structured grid model is termed as the fixed duct because the duct is held fixed in a position while the velocity vectors are used to simulate the angle-of-attack. On the other

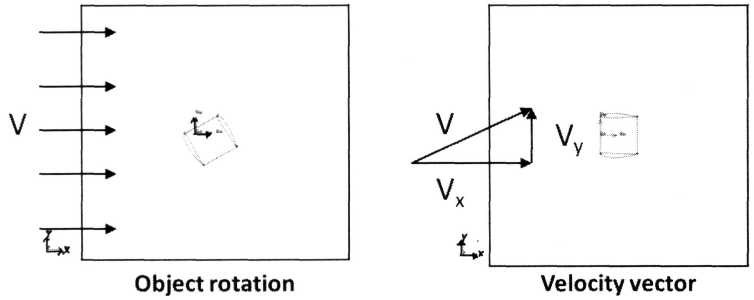

Figure 2: Two Methods to Simulate Angles-of-attack and Sideslip Angles

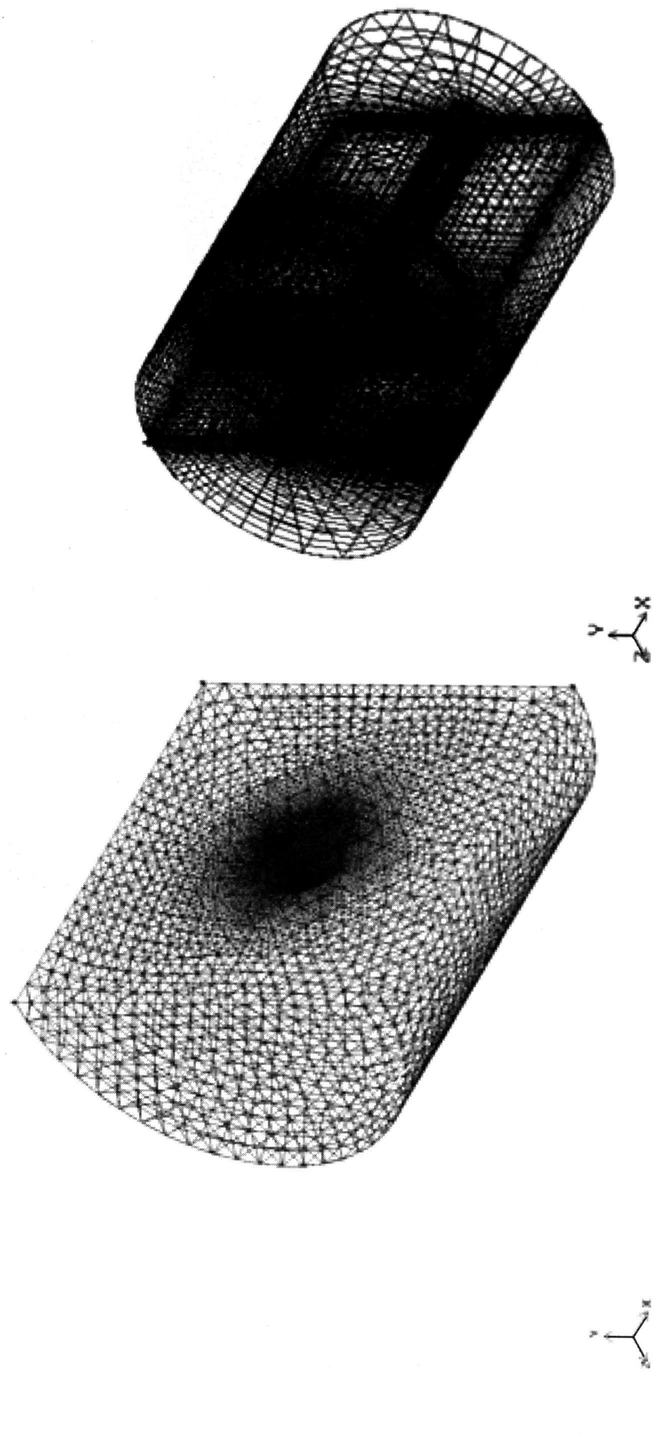

Figure 3: Duct Computational Model: Unstructured (left) and Suctured (right) Grid

hand, the unstructured grid model is termed as the rotated duct because the duct is being rotated in the simulation to represent various angles-of-attack.

The quality of the computational models is tested using a set of tests called the benchmark tests. The objectives of the tests are to study the effect of convergence limit, the faraway boundary distance, the grid independency and the near wall grid distance (wall y+). The benchmark tests results for both fixed and rotated duct models are presented in Table 1.

Table 1: Benchmark Tests Results

Fixed duct	Computational model	Rotated duct
10^{-3}	**Residual criterion** 10^{-3} and 10^{-6}	10^{-6}
1.18	**Grid size (in million of cells)** 0.35, 0.72, 1.18 (structured) 0.5, 1.0, 1,2 (unstructured)	1
3c	**Boundary distance** 3c and 6c	3c
within the recommended range	**Distance to the wall (wall y^+)** wall $y^+<1$ or >30	within the recommended range

Validation Results Discussion

Lift and drag predictions of the fixed and rotated duct models are in a good agreement with the experimental results, particularly at low angles-of-attack (0°-20°). The numerical results are close to the experiment, except at the stall angle-of-attack. Stall region is very difficult to predict due to its complex flow phenomenon. Near the stall angle, the separation point moves rapidly forward and a large wake region is created. This region of turbulent and separated flow is unstable and thus making it difficult to predict. The small values of moment are difficult to be measured accurately in both experiments and computations. However, the solver is still capable of providing reasonably accurate predictions. The moment results for both fixed and rotated duct models show a similar pattern as the experiment (Figure 4).

Based on the confidence in the data accuracy gained after the validation, the solver is used to analyze the interference effects between the duct and the control vane on a VTOL MAV. The simulations are limited to low angles-of-attack (0°-20°) where both models provide the closest agreement with the reference data. For simplicity, the velocity vector method as in the fixed duct model will be used for the interference effects analysis.

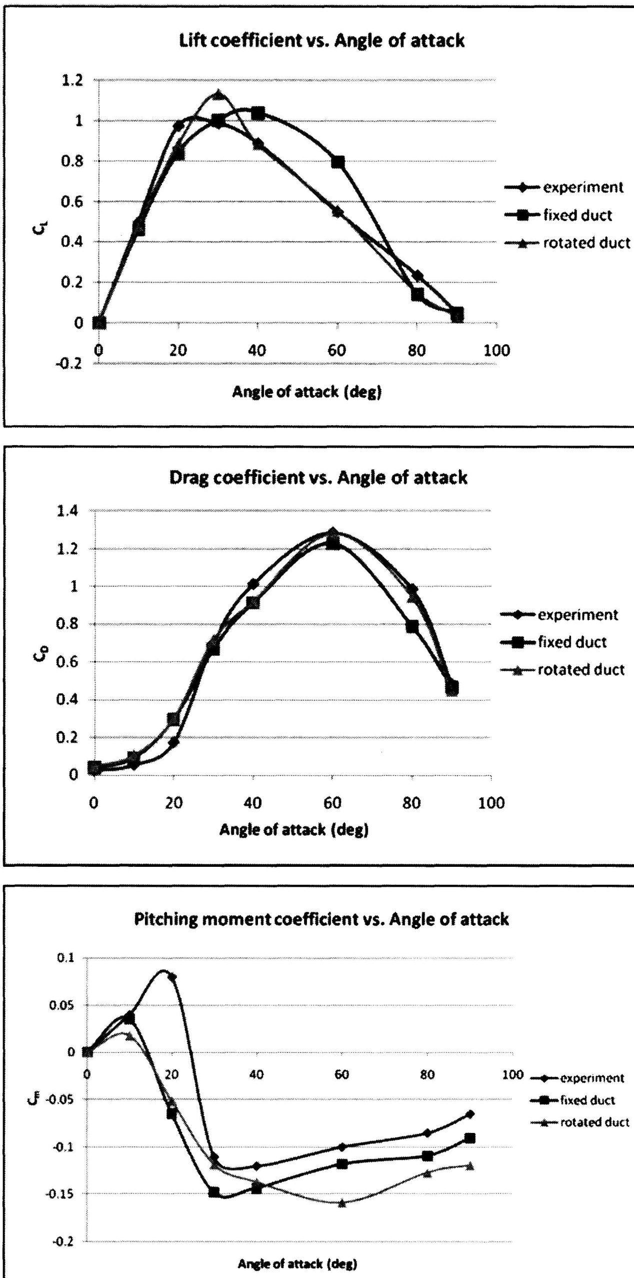

Figure 4: Pitching Moment vs. Angle of Attack for Various Computational Models

Computational Model Descriptions

The investigation of the interference effects between the duct and the control vane is piggybacked on the VTOL MAV design that is being developed. The main components of the VTOL MAV analyzed in this work are ducted-propeller, wing, and control vane, as well as vertical and horizontal tails (Figure 5). The duct has an Eppler-180 airfoil section, a chord length (c) of 0.15 m and an inner diameter (d) of 0.2 m which correspond to an aspect ratio of 1.33. The wing has the same airfoil section as the duct, a span (b) of 0.42 m and an area (S) of 0.076 m^2. The tapered-wing has a low aspect ratio of 2.32, a twist angle -2° and a sweep angle of 5°. The control vane and empennage are consisted of a symmetric airfoil, NACA 0018, with a rectangular platform, 0.06 m chord and 0.08 m span. The control vanes are located exactly at the duct exit. A two-blade propeller is mounted inside the duct. In this research, the MAV is simulated without the propeller due to the assumption that the vane effects on aerodynamic forces and moments are almost independent of thrust coefficient [3].

Based on the setting angle (δ), the computational models of the control vanes can be divided into three categories: symmetrical, asymmetrical-equal and asymmetrical-unequal. Symmetrical control vanes have the same setting angle for the left (δ_L) and the right (δ_R) planes of the vane. The asymmetrical-equal models have the same magnitude of δ but with different sign for the left and right planes of the control vane. The last model is the asymmetrical-unequal with different setting angle for the two planes of the control vane. A total of six configurations are tested for the longitudinal mode analysis, and only five for the lateral-directional. Two configurations for the symmetry category are 0° – 0° and 10° – 10° ($\delta_L - \delta_R$). For the asymmetrical-equal category, two configurations with the setting angles of –5° – 5° and –10° – 10°. Only the latter configuration is tested for the lateral-directional analysis. The last category, the asymmetrical unequal, is represented by two configurations 0° – 5° and 10° – 0°.

The full computational domain is tube-shaped and has a sphere-shaped inner domain which surrounds the object being investigated (Figure 6). The full configuration model of the VTOL MAV consists of around 1.4 million tetrahedral cells. The individual models of the control vanes are meshed using approximately 700,000 tetrahedral cells. These cell numbers are used because they give almost the same, for all models, wall y^+ value of 30, which is within the recommended range [9].

Numerical Test Procedure

Individual and full-configuration computational models are simulated in this research. In the individual analysis, each component of the VTOL MAV is simulated individually and independently without the influence of other

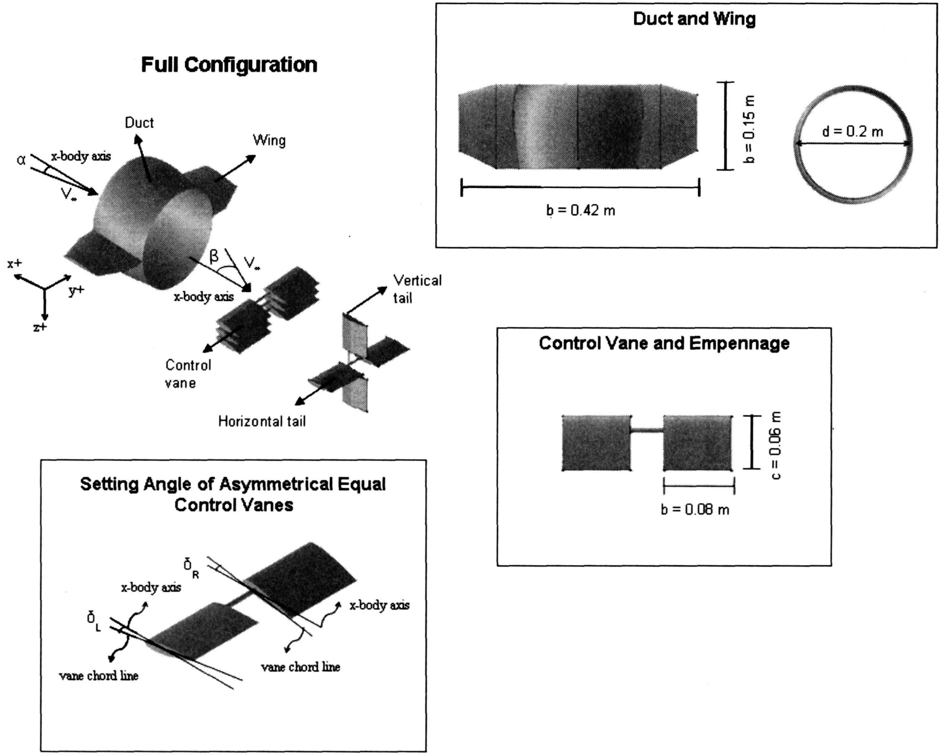

Figure 5: Geometry Details of VTOL MAV

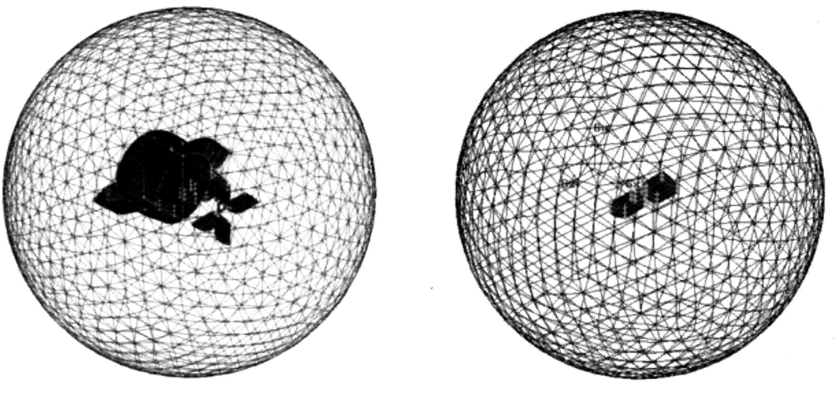

Figure 6: Mesh of Full-configuration VTOL MAV and Individual Control Vane

components. The full configuration analysis includes all the components of the VTOL MAV, such that the flow around a component is perturbed by others. Only the aerodynamic data of the vane are taken from the full configuration analysis. These data are compared to the individual simulation results to analyze the interference effects between the duct and the control vane.

The flow-solver, Fluent, is set with Spalart-Allmaras turbulence model, SIMPLE scheme, and second order upwind discretization. The models are tested for the angles-of-attack (α) of 0°, 10° and 20°. A range of sideslip angles (β) from –10° to 10° with 5° intervals is investigated for the lateral-directional analysis. The vehicle is simulated with freestream velocity of 15 m/s, density of 1.225 kg/m^3 and temperature of 288.15 K. Aerodynamic coefficients are calculated using the wing's span and area as the reference length and area respectively. The moment coefficients are computed with respect to the center of gravity (cg) of the MAV, which is located 0.3709 m from the duct's lip.

Aerodynamic Interference Effects between Duct and Control Vane

In the individual analysis, the control vane is simulated as an isolated component. For that reason, each component in the individual analysis experiences flows with freestream angles-of-attack (α) or freestream sideslip angles (β). The angle-of-attack or sideslip angle of the flow over a component in the full configuration analysis is affected by the interference effects with other components. For analysis purpose, the perturbed angle is marked by "local" subscripts as in local angles-of-attack (α_{local}) or local sideslip angles (β_{local}). The influence of the duct-vane interference effects to the aerodynamic forces and moments of the control vanes are explained next.

Aerodynamic Forces

The computational results of the control vanes show that the lift and drag predictions for the individual simulations/analysis are significantly higher than the full configuration (Figures 7-9). The individual analysis over predicts the lift (C_L) and drag (C_D) coefficients of the control vanes, the asymmetrical-equal -10° - 10° at the angle-of-attack of 10° for example, by more than 250% and 17% respectively with respect to the full configuration analysis results. At 20°, the individual analysis again over predicts C_L and C_D of the same asymmetrical-equal control vane to be more than 200% and 50% higher than the values predicted using the full configuration analysis. This deviation of the individual simulation results from the full configuration ones is found in the aerodynamics forces predictions of the other categories of control vanes. It is also found that the lift and drag curves for the full configuration analysis has significantly lower slopes compared to the individual one.

The notably lower lift and drag coefficients of the control vane in the full configuration analysis are predicted to be caused by the duct located upstream.

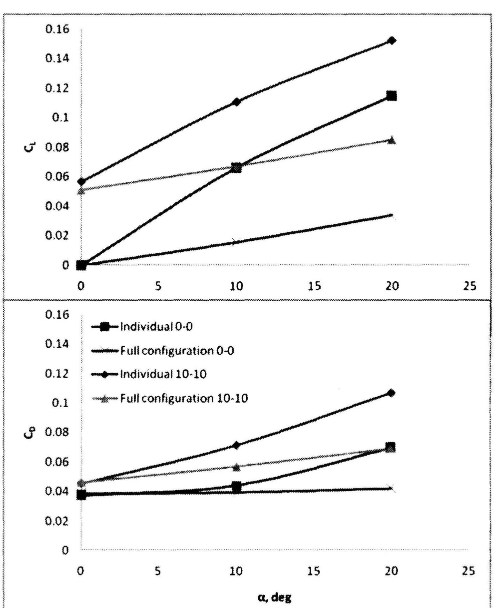

Figure 7: Lift and Drag of Symmetrical Control Vanes

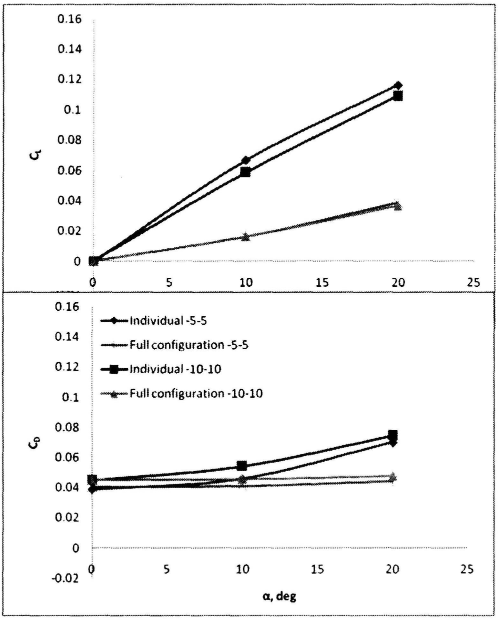

Figure 8: Lift and Drag of Asymmetrical-equal Control Vanes

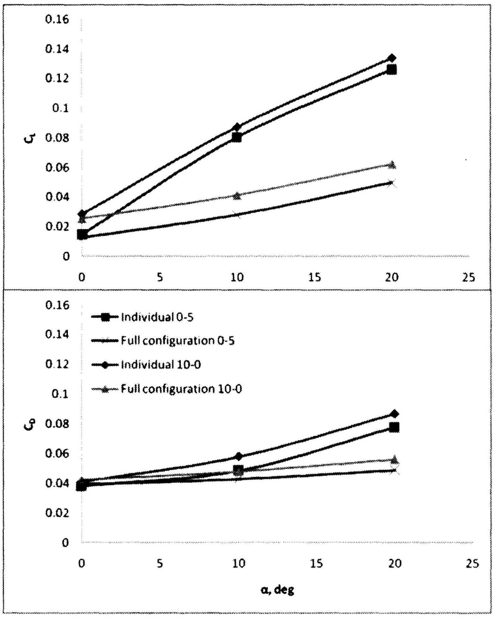

Figure 9: Lift and Drag of Asymmetrical-unequal Control Vanes

The duct has an aligning effect to the vehicle's x-body axis for the flow that passes through it. Therefore, the local flow leaving the duct and entering the control vane is straightened in the direction of the x-body axis, despite of the angle-of-attack of the freestream flow (Figure 10). The flow passes the control vane is subjected to a lower local angle-of-attack compared to the freestream one, thus the lift and the drag coefficients are lower than the values provided by the individual analysis at the freestream angle-of-attack.

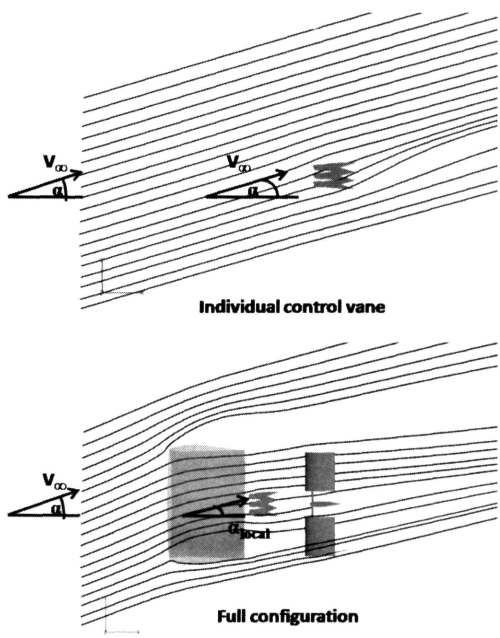

Figure 10: Flow Passes Duct and Control Vane at $\alpha = 20°$ (y-axis plane)

Aerodynamic Moments

Similar to the lift and drag predictions, the rolling and yawing moment predictions of the individual analysis are higher than the full configuration ones (Figures 11 and 12). This over predictions can be clearly seen by comparing the slope of the rolling and yawing moment curves for the individual and full configuration analysis. The curve-slope of the individual analysis is higher than the full configuration, particularly for the yawing moment predictions. The rolling moment of the VTOL MAV does not vary considerably with sideslip angles (β). The vehicle which includes an annular airfoil/duct in its configuration has the

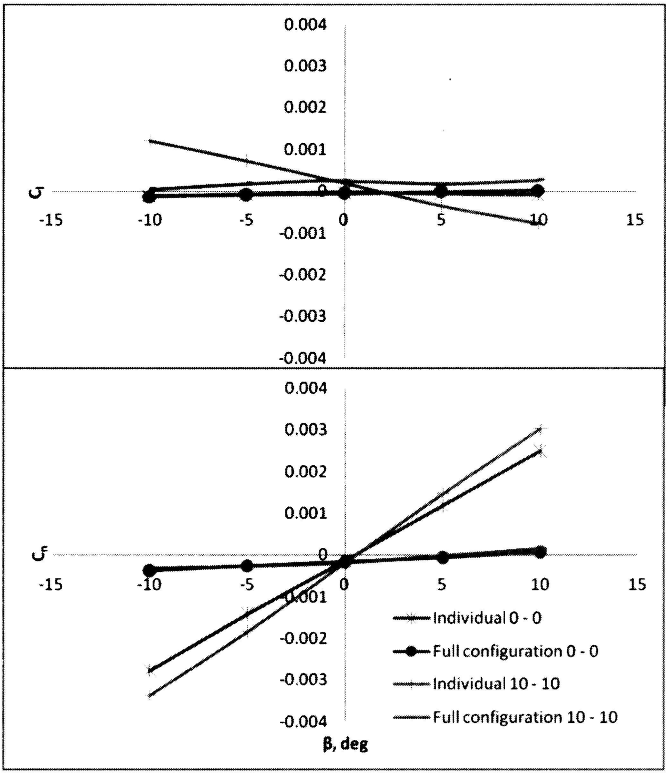

Figure 11: Rolling and Yawing Moments of Symmetrical Control Vanes

tendency to generate a constant rolling moment under various sideslip angles. Nevertheless, the gentler slope of the rolling moment curve for the full configuration simulations is still observable.

The very small aerodynamic moments predicted by the full configuration analysis suggest that the flow coming from sideways, under the influence of sideslip angles; only have a small influence on the control vanes. It is predicted that the vanes are influenced more by the flow coming from the duct outlet. The freestream flow (sideway direction) which enters the duct is straightened in the x-body axis direction of the vehicle. The control vanes experience a local sideslip angle which is much lower than the freestream sideslip angle (Figure 13). As a result, the vanes generate much lower rolling and yawing moments compared to what is predicted by the individual simulations, where the control vane is simulated individually and independently.

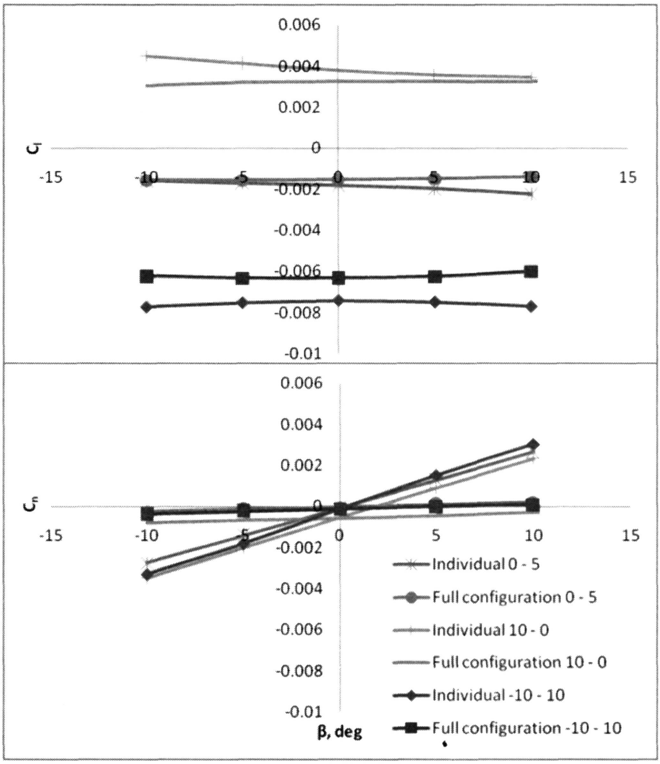

Figure 12: Rolling and Yawing Moments of Asymmetrical Control Vanes

Conclusions

The interference effects between the duct and the control vane have a significant influence on the aerodynamic forces and moments of the VTOL MAV. The duct changes the direction of the freestream flow before it passes the control vane, affecting the flow locally. As a result, the vane's behaviors are dictated by the local flow instead of the freestream one. The differences in the aerodynamic forces and moments of the control vanes, individually and under the influence of the duct, need to be taken into consideration in designing the vehicle. As part of the VTOL MAV, the vanes generate significantly lower aerodynamic forces and moments than as an individual component. If the individual analysis results of the control vanes are used in designing the MAV, the over predictions of the aerodynamic forces and moments will introduce a high error to the design. Furthermore, this error in calculations might lead to a design failure.

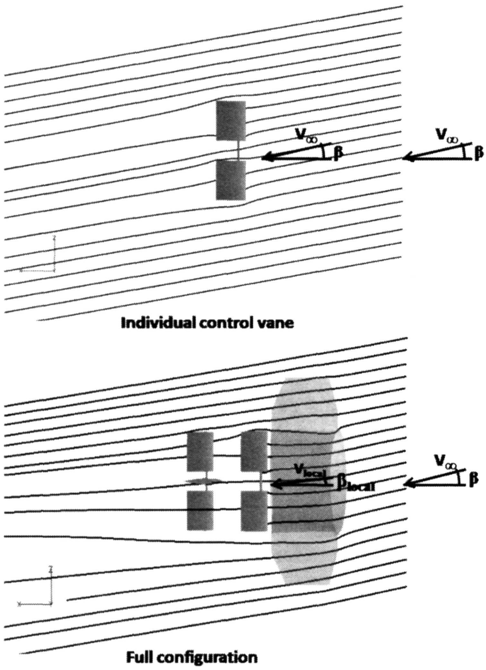

Figure 13: Flow Passes Duct and Control Vane at $\alpha = -10°$ (x-axis plane)

References

[1] Robert, J. and Platt, J. (1948). "Static Tests of A Shrouded and an Unshrouded Propeller," Langley, Virginia, NACA Research Memorandum L7H25.

[2] O. z. Armutcuogˇlu, e. K. g. Mehmet, S. and Tekinalp, O. (March-April 2004). "Tilt Duct Vertical Takeoff and Landing Uninhabited Aerial Vehicle Concept Design Study," *Journal of Aircraft,* vol. 41, 215-223.

[3] Betzina, M. D. and Kita, R. (1980). "Aerodynamic Effects of an Attitude Control Vane on a Tilt-Nacelle V/STOL Propulsion System," *Journal of Aircraft,* vol. 17, 817-822.

[4] Schaefer, C. G. and Baskett, L. J. (2003). "Goldeneye: The Clandestine UAV," in 2^{nd} *AIAA "Unmanned Unlimited" Systems, Technologies, and Operations*, AIAA-2003-6634, San Diego, California.

[5] Guerrero, I., Londenberg, K., Gelhausen, P. and Myklebust, A. (2003). "A Powered Lift Aerodynamic Analysis for the Design of Ducted Fan UAVS," in *2nd AIAA "Unmanned Unlimited" Systems, Technologies and Operations*, AIAA-2003-6567, San Diego, California.

[6] Ko, A., Ohanion, O. J. and Gelhausen, P. (2007). "Ducted Fan UAV Modeling and Simulation in Preliminary Design," in *AIAA Modeling and Simulation Technologies Conference and Exhibit*, AIAA-2007-6375, Hilton Head, South Caroline.

[7] Tobias, E. L. and Horn, J. F. (2008). "Simulation Analysis of the Controllability of a Tandem Ducted Fan Aircraft," in *Atmospheric Flight Mechanics Conference and Exhibit*, AIAA-2008-6700, Honolulu, Hawaii.

[8] Fletcher, H. S. (1957). "Experimental Investigation of Lift, Drag, and Pitching Moment of Five Annular Airfoils," N. A. C. f. Aeronautics, Ed.

[9] Fluent. (2007). "Fluent User Guide Version 6.3," Lebanon, NH, USA: Fluent Inc.

JOURNAL OF MECHANICAL ENGINEERING (JMechE)

Aims & Scope

Journal of Mechanical Engineering (formerly known as Journal of Faculty of Mechanical Engineering), is an international journal which provides a forum for researchers and academicians worldwide to publish the research findings and the educational methods they are engaged in. This Journal acts as a vital link for the mechanical engineering community for rapid dissemination of their academic pursuits as well as a showcase of the research activity of FKM for the outside world.

Contributions are invited from various disciplines that are allied to mechanical engineering. The contributions should be based on original research works. An attempt will be made to review the submitted contributions with competent internal and external reviewers as to the suitability of the paper for satisfying the objectives of the journal.

All papers submitted to JMechE are subjected to a rigorous reviewing process through a worldwide network of specialized and competent referees. To be considered for publication, each paper should have at least two positive referee's assessments.

General Instructions

Manuscripts intended for publication in the Journal FKM should be written in camera ready form with production-quality figures and done electronically in Microsoft Word 2000 (or above) and submitted along with one hard copy. Manuscripts should be typed using Times New Roman font (point 10) on one side of the paper (A4 size), single-spaced, with wide margins (1 in left and right, and 1 in on top and bottom). The manuscript should include the title of the paper; the author's name and affiliation; a short abstract of between 200 and 300 words which clearly summarises the paper; and a list of keywords. Limit your submission to a maximum of 20 typed pages.

Keywords

Keywords supplied by the author should appear on a line following the abstract. The keywords selected should be comprehensive and subject specific. Maximum of five keywords should be sufficient to cover the major subjects of a given paper. General terms should not appear as keywords, as they have little use as information retrieval tools. Please choose keywords to be as specific as possible and list the most specific first, proceeding to the most general last.

Units

All scientific and technical data presented should be stated in SI units.

Footnotes

Footnotes should be kept to an absolute minimum and used only when essential.

Formulae

Formulae should be typewritten using MS Word compatible Equation Editor.

Tables

Tables, should be included within the text where appropriate and must be numbered consecutively with Arabic numerals and have titles that precede the table. They should be prepared in such a manner that no break is necessary.

Figures

Authors should appreciate the importance of good-quality illustrations. All graphs and diagrams should be referred to, for example, Figure 1 in the text. All figures must be numbered consecutively with Arabic numerals. A detailed caption should be provided below each figure according to the following format:

Figure 1: (a) A simple 2-D cantilever and (b) microcantilever with a diamond probe.

Figures should be embedded within the text where appropriate. Glossy photographs when required should be scanned to a resolution suitable with the reproduction requirements (1200 dpi generally will be sufficient).

References

Use squared brackets to indicate reference citation such as [1], [3]-[5] in the main text. Include references at the end of the paper according to the citations order that appears in the paper using the following format.

[1] M. K. Ghosh and A. Nagraj, "Turbulence flow in bearings," Proceedings of the Institution of Mechanical Engineers 218 (1), 61 - 64 (2004).

[2] H. Coelho and L. M. Pereira, "Automated reasoning in geometry theorem proving with Prolog," J. Automated Reasoning 2 (3), 329-390 (1986).

[3] P. N. Rao, Manufacturing Technology Foundry, Forming and Welding, 2nd ed. (McGraw Hill, Singapore, 2000), pp. 53 – 68.

[4] Hutchinson, F. David and M. Ahmed, U.S. Patent No. 6,912,127 (28 June 2005).

All papers must be submitted online to jmeche.int@gmail.com

Correspondence Address:

Editor in Chief
Journal of Mechanical Engineering (JMechE)
Faculty of Mechanical Engineering
Universiti Teknologi MARA
40450 Shah Alam, Malaysia.

Tel : 603 – 5543 5161
Fax : 603 – 5543 5160
Email: jmeche.int@gmail.com