Proceedings of the ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting and 8th International Conference on Nanochannels, Microchannels, and Minichannels
FEDSM-ICNMM2010
August 1-5, 2010, Montreal, Canada

FEDSM-ICNMM2010-30795

ANALYZING THE PERFORMANCE OF A HOVERING DUCTED ROTOR IN GROUND/WALL EFFECTS TO IMPROVE THE CONTROLLING ASPECTS OF VTOL VEHICLES IN CONFINED SPACES

Zahra Hosseini
Department of Mechanical and Manufacturing Engineering,
University of Calgary
Calgary, Alberta, Canada
Email: zhossein@ucalgary.ca

Robert J. Martinuzzi
Department of Mechanical and Manufacturing Engineering,
University of Calgary
Calgary, Alberta, Canada
rmartinu@ucalgary.ca

Alejandro Ramirez Serrano
Department of Mechanical and Manufacturing Engineering,
University of Calgary
Calgary, Alberta, Canada
aramirez@ucalgary.ca

ABSTRACT

This paper examines ground/wall effects on a ducted fan, with the aim to improve the handling quality of an unmanned aerial vehicle targeted for operations in confined spaces. In order to do this, flow field at different configurations of the fan-ground-wall is visualized computationally in Fluent. Herein the standard k-ε model and the non-equilibrium wall function are used to model the turbulence while the fan is modeled using the Virtual Blade Model (VBM). It is shown that in ground effects thrust is larger at distances relatively close to the ground. Adding a wall introduces an asymmetry to the flow which produces a considerable amount of horizontal force attracting the fan towards the wall. Moreover, in presence of the wall, the total thrust reduces slightly. The paper also describes the observation that at sufficiently close distances between the fan and the wall/ground, a large portion of the downwash re-enters to the fan which causes significant performance loss.

INTRODUCTION

There are a number of Vertical Take-Off and Landing (VTOL), manned and unmanned, vehicles that are used in a number of operations. Despite their omni-directional and high maneuver characteristics these vehicles have not been used in confined environments where their characteristics would enhance the execution of complex tasks such as search and rescue within collapsed building. The eVader is a novel VTOL aerial vehicle which benefits from a dual lifted-fan Oblique Active Tilting (dOAT) concept design, which allows it to move within helicopter impenetrable environments. This design, based on the gyroscopic effects principle, produces large moments and eliminates the necessity of large moment arms, such as ailerons, found in traditional aircrafts [1], thus making possible move within confined spaces and executing maneuvers that current aircrafts cannot execute (e.g., pitch hover). This highly maneuverable and compact design concept makes the eVader a special alternative for urban operations such as search and rescue, and aerial surveillance, just to name a few. Although the eVader is designed to be agile enough to fly among urban obstacles, the controlling quality is significantly affected by the changes in rotor performance in proximity of solid obstacles (e.g., walls). To have a successful flight these changes must be characterized and taken into account in the vehicle's control. This paper aims to examine the ground/wall effects on produced forces of a ducted rotor hovering in proximity of walls and grounds (vertical and horizontal surfaces).

The ground/wall effects analysis can also be important to understand the *burnout* phenomenon. *Burnout* happens when the flow velocities in the wall jet reach a high value which may lift up debris and reduce the pilot's visual domain. In order to understand and predict this phenomenon, the flow profiles close to the ground/wall must be examined. Beside force changes, analysis of flow induced by the fan, *downwash*, is presented in this paper.

The downwash of a rotor in ground effects is very similar to that of an impinging jet. The impinging jet flow has three regions [2]: free jet, deflection and wall jet. In the first region, the flow approaches the ground, and therefore is subjected to normal straining. Subsequently the ground deflects the flow laterally creating an axisymmetric wall jet. The wall jet region is expected to satisfy the well known similarity regions [3], in the absence of lateral walls, and scale with the maximum local velocity, U_m, and the wall jet thickness, δ, which is defined as the height where the velocity is $U_m/2$. The flow structure of an axisymmetric impinging jet is shown in Figure 1.

The difference between the rotor's downwash and that of the impinging jet is the presence of tip vortices and swirling. In the considered case, the duct prevents the generation of tip vortices. Moreover, it is shown that the swirling velocities are negligible. Therefore, in the absence of the lateral wall, the flow characteristics are expected to be similar to those of the impinging jets.

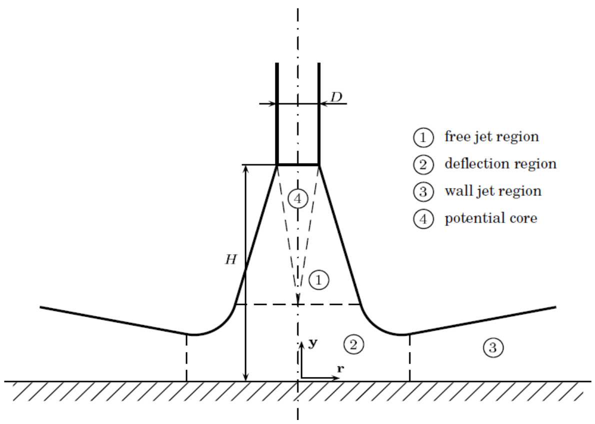

FIGURE 1. THE REGIONS OF AN IMPINGING JET [2].

Numerous researchers have addressed the significant changes in the performance of a rotor flying in ground effects (IGE). The ground effects in [4, 5] are formulated as a modification to the required power by the method of images. In [6] an empirical model is proposed to estimate the change of the required power for a helicopter in ground effects based on flight data of numerous flight tests. These studies have reached acceptable results by applying simple methods but are restricted to the situation where the ground is relatively far and lateral boundaries are absent (e.g. walls). Therefore more accurate models, such as solving Navier Stokes equations, could yield better understanding of both required power and flow field patterns.

There are a number of other studies that have examined ground effects in more detail [7-10]. In [7] the flow visualization of a jet powered VTOL flying IGE is conducted experimentally, and in [8] a full scale helicopter operating IGE is visualized numerically. These two papers discussed the ground effects qualitatively. Examining the flow field qualitatively gives a general view of flow features in ground effects; however, a quantitative approach is also necessary to estimate the force and moment coefficients required by the vehicle's navigation system.

In [9] the Euler equations are solved to simulate a four blade rotor hovering at different heights above the ground. The authors in [9] used a moving overlapped mesh (i.e., a high density grid around the blades and a uniform Cartesian grid on the background) while the flow is interpolated in the boundaries between two grids. Comparisons of their analytical results with experimental data showed a large difference at high distances of the fan above the ground due to the fact that the viscous effects are neglected and the boundary layer is predicted poorly.

Experimental visualization of a hovering rotor at different heights above the ground is presented in [10] where the authors studied the velocity profiles in the wall jet and thrust. The studies presented in [10] are used in this paper as a mean to verify the numerical simulations.

Modeling an individual rotor in detail requires significant computational resources. To reduce this cost, the Virtual Blade Model (VBM) has been a very common approach (e.g., [8] and [11]). In this model, the rotor is modeled as a thin disc which reduces grid requirements significantly. As described in [12] this model is simple and practical to use, especially for complex flows, such as the one being considered in this paper.

To model the fan by VBM two alternatives can be employed [11]: momentum disc and pressure disc. In the first approach, aerodynamic forces on the blades are calculated, by using the available blade element theory for example, and the effects of the generated forces are added to the momentum equation as source terms. In the second approach, the disc is considered as an actuator disc which introduces a pressure rise across the disc. Here the second approach is used since it is easier to employ and requires less computational resources, while yields acceptable results by using a reasonable prescribed boundary condition.

Despite many studies conducted for ground effects, for wall effects little information is available in the literature. In [13] a bird flying at the center of a box is modeled by using method of images and an approximation for the required power is obtained. This approach cannot account for viscous effects. In our paper one lateral wall is modeled. It is expected that understanding the flow features of this case, will help us to predict the flow features in more complicated cases, such as flying in confined spaces while potentially using one or more ducts included in the vehicle design.

In this paper, FLUENT (version 6.3.26) is used to simulate and analyze the ground/wall effects on a hovering ducted rotor at different configurations. For this, first the rotor at different heights above the ground is simulated followed by adding a vertical wall to the geometry. Flow patterns and aerodynamic forces are analyzed and obtained results are presented.

To examine the accuracy of the simulations, the nondimensional velocity profiles in the wall jet as well as the changes in the thrust are compared with the available information about impinging jets. These comparisons can also show the reliability of the predicted wall effects, since the same approach is employed for the wall simulations. In addition to this, the results of the wall simulations are compared with flows which have similar features. A helicopter landing on a ship [14], or flying with a low forward speed close to the ground [15], has common features with a fan hovering in the proximity of ground/wall, in view of flow pattern. In all these cases, a recirculation is developed at one side of the fan, which draws back the downwash to the fan. Due to this recirculation the inflow and consequently the required power increases. Generally, the comparisons show good agreements with available information in the literature.

METHODOLOGY

The incompressible Navier Stokes equations are employed to solve the flow. To model the turbulence far from the boundary layer, the standard k-ε approach is applied [16]. To match the flow in the turbulent core with the flow in the boundary layer, the non-equilibrium wall function presented in [17] is used. This model is a modification to the standard wall function which takes into account the pressure gradient. In the problem at hand, the pressure gradient under the fan and close to the ground is large. Thus this model is more accurate when compared with the standard wall function. To discretize the governing equations, a second order scheme was adopted.

To analyze the ground effects, a 2-D axisymmetric geometry with respect to the fan axis is considered. However in the presence of a wall, the flow is no longer axisymmetric and the 2-D geometry cannot be used. Instead, a symmetric 3-D domain is used for wall simulations. These two domains are shown in Figure 2.

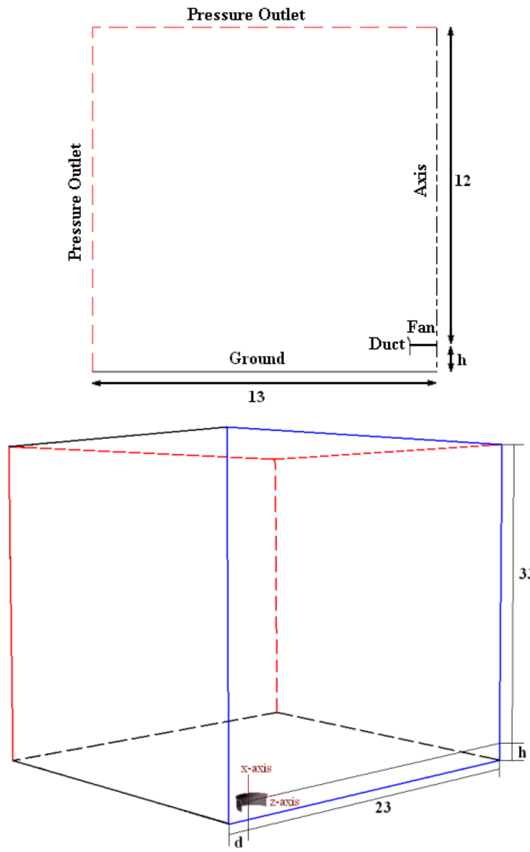

FIGURE 2. THE 2-D AND 3-D DOMAINS USED IN THE SIMULATIONS.

The dimension of the 2-D grid used is 13r×(12+h)r (~60,000 cells) and the 3-D grid is 23r×(23+d)r×(33+h)r (~15,000,000 cells), where h is the rotor height above the ground, d is the rotor distance from the wall and r is the radius of the rotor.

The no slip condition is used as the boundary condition at the solid boundaries. The outer boundaries are assumed far enough such that the static pressure is considered to be ambient. For all other variables used in the model, the Neumann boundary condition is used.

In order to model the fan, the VBM is used. The VBM requires that the pressure jump and tangential velocity be prescribed. In this paper, the pressure jump condition on the fan based on the rotors used in our vehicle (i.e. 16x10 in Windsor propeller, true pitch, NACA airfoil) is defined by the fan curve (i.e., Poly curve) displayed in Figure 3.

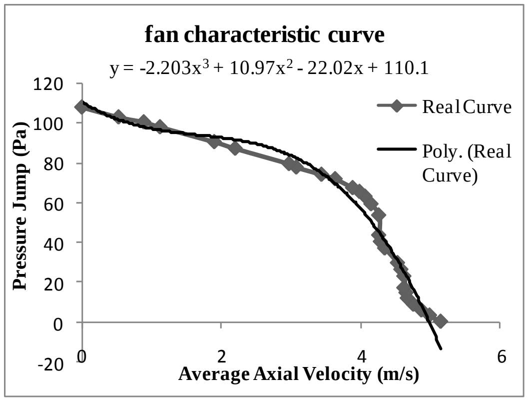

FIGURE 3. FAN CHARACTERISTIC CURVE USED AS THE BOUNDARY CONDITION ON THE FAN.

The curve relates the pressure jump to the average normal velocity on the fan. This curve is a scaled version of a real fan curve [18] while the curve axes are scaled to match the desired conditions present in our vehicle prototype (i.e., eVader). Therefore, the results of the simulations are expected to show a reasonable trend. Regarding the need to approximate the tangential velocity, the pressure rise equation across an ideal compressor stage [19] can be used.

To show that the solution is not affected by the shape of the domain, the simulations of the ground effects were performed on cylindrical and hemispherical domains with radius of 9r, 13r and 26r. In all such initial simulations, the fan was placed 1r above the ground and the power law scheme was used.

The comparisons between results for the six domains showed that in order to obtain the same solution accuracy the hemispherical domains need to be larger than the cylindrical domains. The other distinction between these two domains is that while most of the cells in the cylindrical domain are rectangular, a large number of cells in the hemispherical domain are highly skewed.

The results of the 13r and the 26r domains were very close to each other and in general it can be said that the results are in agreement. Therefore, the cylindrical domain with radius of 13r was used in the final simulations.

The second step in the grid study was to check the influence of grid cell density. Simulations were conducted on 2D grids of sizes 115x130, 230x260, 460x520 and 920x1040. The comparisons of the results on these grid sizes show convergence. In Figure 4, for example, the differences in axial velocity and thrust due to momentum are shown. Note that labels 1, 2, 3 and 4 refer the finest grid to the coarsest respectively. The computational time for the finer grids was much lengthier (3 and 7 times more). This would be a more critical issue in the wall effects (3-D) simulations as the number of cells increases by two to power of three (2 million cells vs. 16 and 128 million cells). Therefore it was decided to use the 230x260 grid. However, it was observed that at some points the errors of the solution on this grid are rather large. To reduce these errors a more accurate method (second order scheme) was employed and as a result the solution on the 230x260 grid got closer to the 460x520 grid. In Figure 4 for example, the difference between the momentum thrusts between these two grids are shown. Therefore, the 230x260 with the second order scheme was used in the simulations.

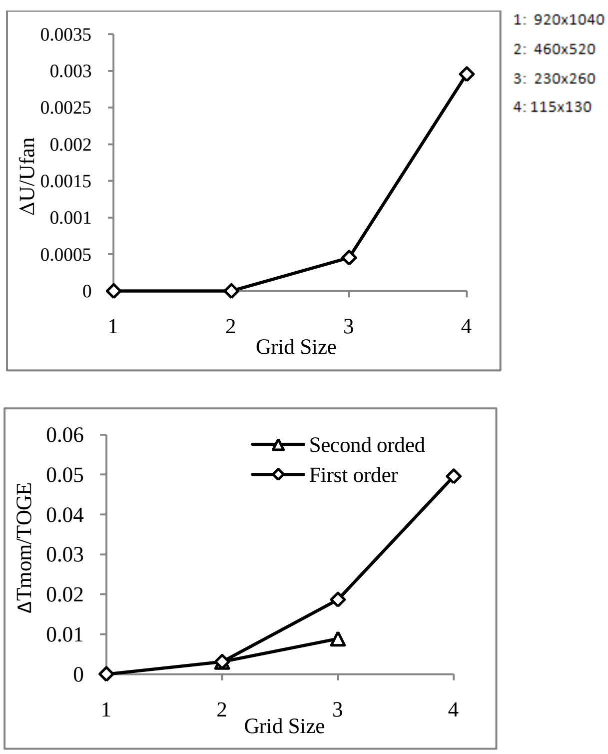

FIGURE 4. THE DIFFERENCES IN THE MOMENTUM THRUST (N) AND THE AXIAL VELOCITY (M/S) AT ONE OF THE POINTS IN THE FOUR DOMAINS CONSIDERED FOR CELL DENSITY STUDY.

To simulate the flow in the presence of a wall, a 3-D grid based on the results of the 2-D domain was constructed. The independency of results of the ground effects simulations were checked, by performing the same simulations on this 3-D domain and comparing the obtained solution with the 2-D results.

For the wall effects simulation, the grid study on the obtained 3-D domain was performed by simulating the flow on domains with different dimensions. The comparisons of the flow parameters show that in order to have a flow independent of domain, the dimensions of the grid in the presence of a wall (23r×23r×33r) must be larger than the grid used in ground effect simulations (13r×13r×13r). It was also observed that the grid size close to the fan, where the gradients are high, must be finer. Therefore, it was decided to use twice finer cells close to the fan, and the same cell sizes for the rest of the domain.

RESULTS AND DISCUSSION

To analyze the ground effects, the fan at eight heights above the ground was simulated: 0.7r, 1r, 1.5r, 2r, 2.5r, 3r, 4r and 4.5r. To examine the effects of the lateral wall, a wall was added at distances of 2r and 4r from the fan in the first two simulations of the ground effects.

In order to examine the effect when including a tangential velocity on the fan, a simulation was performed for a fan at height of 1r above the ground with a prescribed constant tangential velocity. The magnitude of the tangential velocity on the fan was estimated using the relation between the pressure rise and the tangential velocity for an ideal compressor [19]. As expected, the ratio of the tangential velocity to the average axial velocity on the fan was very small (~2%); and the comparison of the flow parameters in the simulations with and without swirl showed that the swirl does not affect the flow noticeably.

The simulations of the ground effects show an increasing trend of the thrust as the fan becomes closer to the ground. Thrust is composed of two parts: thrusts due to momentum and pressure. At lower heights the thrust due to momentum decreases since the fan operates at lower speed on its characteristic curve (Figure 5). However, the thrust due to pressure increases since a high pressure region is developed under the fan. The components of the thrust and the total thrust obtained in our study are plotted in Figure 6. As it is observed from Figure 6, the pressure thrust is larger than that of momentum; as a result the total thrust increases at lower elevations. Another point to notice is that the thrust does not change significantly at heights greater than 2r.

In the work reported in [10], a set of experiments was arranged to visualize the flow around an isolated fan at different heights above the ground, up to a height of 3r. The thrusts presented in [10] shows the same trend as the results obtained in our simulations (i.e., the closer is the distance of the fan to the ground the larger is the thrust; and no significant change at heights larger than the diameter of the fan).

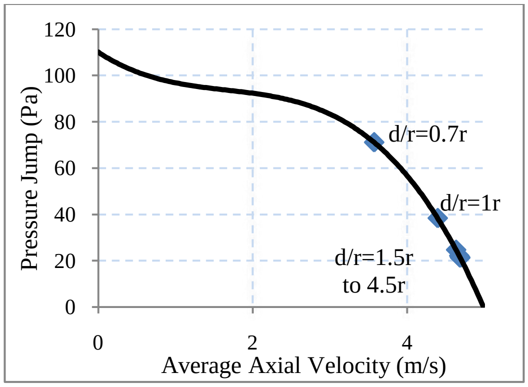

FIGURE 5. THE FAN OPERATING POINTS AT DIFFEREFT ELEVATIONS.

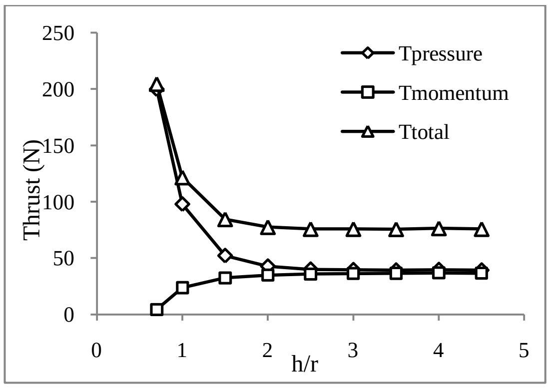

FIGURE 6. THRUST COMPONENTS AND THE TOTAL THRUST AT DIFFERENT ELEVATIONS.

The normalized radial velocities for the separation of 2r from the ground are plotted in Figure 7. The velocity is normalized by the local maximum velocity, U_m, and the distance from the ground (y) by the jet thickness δ. It can be seen than, except for $x/r = 2$, all the profiles coincide. This shows the self similarity of the profiles [3]. It also suggests that the impingement region is located at $x/r < 3$.

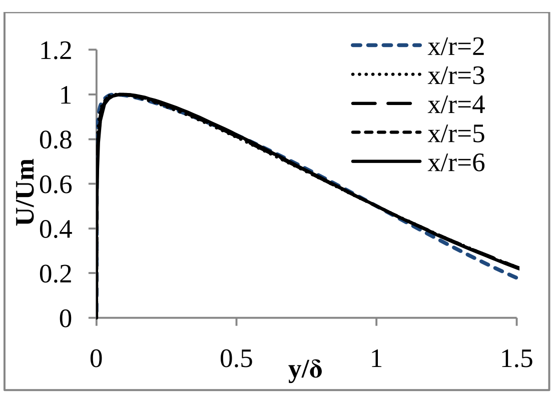

FIGURE 7. THE NORMALIZED RADIAL VELOCITY AT DIFFERENT RADIAL POSITIONS.

From the simulations of the wall effects, it is observed that a recirculation zone is developed between the fan and the wall (Figure 8). This recirculation impacts the fan performances in different ways which are mentioned below.

Because of this recirculation, a large portion of the rotor's downwash re-enters to the rotor. The re-entered flow may carry loose material captured from the surfaces with the potential to cause significant performance losses as well as detrimental damages on the rotor and the fuselage. Moreover, burnout will be a more critical issue under these conditions.

Simulations were performed to estimate the portion of the downwash re-entering to the rotor for two cases (h/r=1, d/r=2 and h/r=0.7, d/r=2). Virtual particles with properties similar to air were added to the air below the fan by a source with a small production rate. As a result, the virtual particles do not interact with the air and do not interfere with the flow field. The fraction of the re-entered flow can be estimated by calculating the ratio of particle concentrations above and below the fan. The results showed that for the first (h/r=1) and second simulations (h/r=0.7) about 25% and about 20% of the downwash re-enters to the fan, respectively. Thus, as the fan gets closer to the ground while in close proximity to a wall the amount of particles that re-enter the fan reduces (but still significant) since the fan is operating at lower speeds on its characteristic curve.

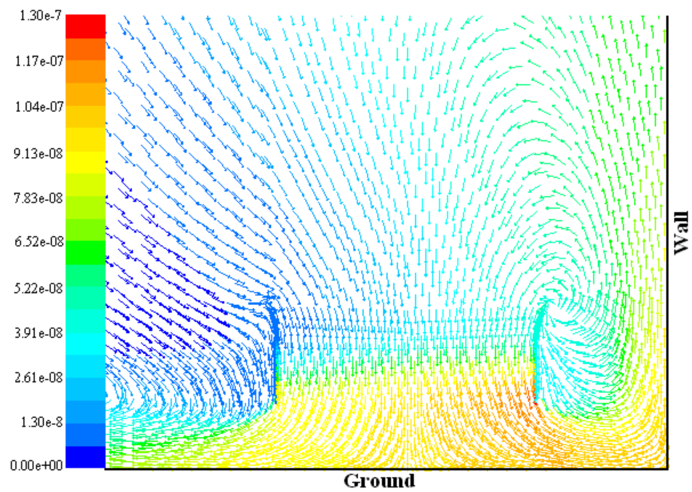

FIGURE 8. CONTOUR OF MASS FRACTION OF THE VIRTUAL PARTICLES ADDED BY SOURCE TEMRS BELOW THE FAN.

The recirculation affects the thrust as well. It increases the inflow to the fan and thus the momentum thrust reduces. Also, it blows away the flow blocked in the high pressure region and reduces the pressure under the fan. Consequently, the pressure thrust at separation of 2r from the wall is smaller than the larger separation (4r). The thrust components and the total thrust are presented in Table 1.

Totally, as the fan approaches the wall, the thrust reduces. In other words, to keep the thrust constant, a larger power is required. In this stage, a quantitative comparison is not possible since enough information is not available, but qualitatively, the changes are reasonable.

TABLE 1. FORCES PRODUCED BY THE FAN AT DIFFERENT CONFIGURATIONS.

	T_{mom} (N)	T_P (N)	T_{tot}(N)	F_z(N)	M_y(N.m)
h/r=1	26.180	94.395	120.575	0	0
h/r=1, d/r=4	18.351	100.954	119.305	-6.012	-2.164
h/r=1, d/r=2	16.292	97.499	113.791	-14.923	-3.970
h/r=0.7	10.639	199.711	210.350	0	0
h/r=0.7, d/r=4	0.443	208.986	209.429	-5.058	-1.670
h/r=0.7, d/r=2	-2.442	207.676	205.234	-14.491	-4.350

The other effect of the lateral wall is the asymmetry that it introduces to the flow; therefore, the net force exerted on the fan has a horizontal component (Table 1). The magnitude of the horizontal force, in sufficiently close distances of the wall, can be fairly noticeable (more than 10% of the thrust). This force attracts the fan towards the wall and finds larger values as the fan approaches the wall. In layman's terms: the wall sucks the fan as the separation is reduced.

The asymmetry in the flow also causes a non-zero moment around the y-axis (pointing outward the page in Figure 8). For instance the moment on the duct in the last simulation presented in Table 1 (i.e., h/r=0.7, d/r=2) is about -4 N.m. To have a safe and stable flight the control mechanisms maneuvering the vehicle in close proximity to the wall must respond appropriately to these changes.

CONCLUSION

Simulation of a ducted fan in proximity of ground/wall using VBM was performed. It was shown that the effect of ground is negligible when the distance of fan from the ground is greater than 2r. An increasing trend for the thrust was observed as the fan moves from the distance of 2r to 0.7r.

In presence of the wall, a recirculation is developed between the fan and the wall which re-enters a large portion of the fan downwash to the fan with the potential to cause significant performance losses. It was also shown that, larger power is required to produce the same thrust of that of a fan hovering IGE. Also because of the asymmetry in the flow, noticeable moment and horizontal force are produced which require an appropriate controlling response.

REFERENCES

[1] Gress, G., 2003, "A Dual-Fan VTOL Aircraft Using Opposed Lateral Tilting for Pitch Control", American Helicopter Society 59th Annual Forum, Phoenix, Arizona.

[2] Hadziabdic, M., and Hanjalic, K., 2006, "LES of Flow and Heat Transfer in a Round Impinging Jet", Springer Netherlands, pp. 477-486.

[3] Glauert, M. B., 1957, "The wall jet", Journal of Fluid Mechanics, 1, pp. 625-643.

[4] Knight, M., and Hefner, R. A., 1941, "Analysis of ground effect on the lifting airscrew", Technical Note no. 835, NACA, Washington.

[5] Cheeseman, I. C., and Bennett, W. E., 1955, "The Effect of the ground on a Helicopter Rotor in Forward Flight", technical report no. 3021, ARCR & M.

[6] Hayden, J., 1976, "The effect of the ground on helicopter hovering power required", 32nd Annual National V/STOL Forum American Helicopter Society, Washington DC.

[7] Mourtos, N. J., Couilaud, S., Carter, D., Hange, C., Wardwell, D., and Margason, R. J., 1995, "Flow Visualization Studies of VTOL Aircraft Models During Hover In Ground Effects", NASA, California.

[8] Modha, A. N., Blaylock, T. A., and Chan, W. Y. F., 2007, "Brown-out – Flow Visualisation using VBM", International Aerospace CFD Conference, Paris.

[9] Tanabe, Y., Saito, S., Ooyama, N., and Hiraoka, K., 2009, "Investigation of the Downwash Induced by Rotary Wings in Ground Effects", Transactions of the Japan Society for Aeronautical and Space Sciences, 10, pp. 20 –29.

[10] Lee, T. E., Leishman, J. G., and Ramasamy, M., 2008, "Fluid dynamics of interacting blade tip vortices with a ground plane", American Helicopter Society 64th Annual Forum, **2**, pp. 1231-1248, Montreal.

[11] Ruith, M. R., 2005, "Unstructured, Multiplex Rotor Source Model with Thrust and Moment Trimming- Fluent's VBM Model", 23rd AIAA Applied Aerodynamics Conference, Toronto.

[12] Lighthill, J., 1979, "A simple fluid-flow model of ground effect on hovering", J. of Fluid Mechanics, 93(4):, pp. 783–797.

[13] Rayner, J. M. V., Thomas, A. L. R., 1991, "On the Vortex Wake of an Animal Flying in a Confined Volume", Philosophical Transactions: Biological Sciences, 334, pp. 107-117.

[14] Wilkinson, C. H., Zan, S. J., Gilbert, N. E., Funk, J. D., 1998, "Modelling and simulation of ship air wakes for helicopter operations-A collaborative venture", RTO AVT Symposium on Fluid Dynamics Problems of Vehicles Operating near or in the Air-Sea Interface", Amsterdam.

[15] Leishmane, J. G., 2006, *Principles of Helicopter Aerodynamics*, Cambridge University Press, New York.

[16] Launder, B. E., and Spalding, D. B., 1972, *Lectures in Mathematical Models of Turbulence*, Academic Press, London, England.

[17] Kim, S. E., and Choudhury, D., 1995, "A Near-Wall Treatment Using Wall Functions Sensitized to Pressure Gradient. Separated and Complex Flows", ASME FED, 217, pp. 273-279.

[18] Stein, J., and Hydeman, M. M., 2004, "Development and Testing of the Characteristic Curve Fan Model", ASHRAE transactions, pp. 347-356.

[19] Dixon, S. L., 1998, *Fluid Mechanics and Thermodynamics of Turbomachinery*, Pergamon Press Ltd., Burlington, MA, chapter5.

