Two dimensional numerical analysis of aerodynamic characteristics for rotating cylinder on concentrated air flow

M S Alias, A S Mohd Rafie*, O F Marzuki, M F Abdul Hamid and C C Chia

Department of Aerospace Engineering, Faculty of Engineering, Universiti Putra Malaysia, Serdang, Selangor, Malaysia.

*shakrine@upm.edu.my

Abstract. Over the years, many studies have demonstrated the feasibility of the Magnus effect on spinning cylinder to improve lift production, which can be much higher than the traditional airfoil shape. With this characteristic, spinning cylinder might be used as a lifting device for short take-off distance aircraft or unmanned aerial vehicle (UAV). Nonetheless, there is still a gap in research to explain the use of spinning cylinder as a good lifting device. Computational method is used for this study to analyse the Magnus effect, in which two-dimensional finite element numerical analysis method is applied using ANSYS FLUENT software to examine the coefficients of lift and drag, and to investigate the flow field around the rotating cylinder surface body. Cylinder size of 30mm is chosen and several configurations in steady and concentrated air flows have been evaluated. All in all, it can be concluded that, with the right configuration of the concentrated air flow setup, the rotating cylinder can be used as a lifting device for very short take-off since it can produce very high coefficient of lift (2.5 times higher) compared with steady air flow configuration.

1. Introduction

Magnus effect can be seen as a device that produces lifting force in perpendicular direction of the flow by providing a moving wall on its body to influence the boundary layer of fluid around it [1]. Several inventions have proven its potential applications in aerospace including the Magnus rotor wing aircraft in the early 1920s but high cost has drastically hindered its development. In the meantime, researchers continue to develop the fan-wing aircraft, which increases the lift coefficient but has some difficulties in low-speed manoeuvring and hard landing [2]. A smaller-scaled rotor wing aircraft design that uses the concept of Magnus rotor wing has been developed to prove the feasibility of flying using Magnus effect principle. The results are promising for take-off and landing in short distance, and the aircraft is able to fly at low speeds and stable at higher angles of attack [3]. Furthermore, another study involves changing the airfoil leading edge into a rotating cylinder for Moving Surface Boundary Layer Control (MSBC) and the result shows an increase in lift by 100%, stall delay from 10° to 35° and increase in lift-to-drag ratio by 167% [4]. In addition, the effect of Magnus rotor to micro air vehicle has also been successfully experimented and it has been found that the size of cylinder and payload is an important factor that contributes to the lifting of the MAV [5].

The flow dynamic of a rotating cylinder will be dramatically altered when the body is placed close to the wall. Results from a previous study show that the lift force experienced by the cylinder has been directed away from the static wall but when the gap ratio is maintained, the lift and drag increase as the
Reynolds number, $Re$ increases [6, 7]. Therefore, a rotating cylinder with no wall close to the body can be a solution to create lift perpendicular to the airstream direction. A different study is carried out to investigate local flow field and rear wake of a surface-mounted finite circular cylinder in low speed wind tunnel using Particle Image Velocimetry (PIV), with $Re = 4.2 \times 10^4$ and different aspect ratios (ARs). The result has concluded that higher AR might increase turbulence and upwash that are formed with flow separation occurred at the leading edge free-end surface and a mean recirculation formation [8]. Furthermore, separation point is identified on the basis of the highest pressure coefficient on the cylinder surface [9]. Stagnation is considered as high pressure due to the existence of turbulent flow. The cylinder tends to move in y-direction due to pressure created near to the lower side of the cylinder symmetry. This is based on specific $Re$, rotational rate and angle of incidence of the air flow [10]. On the other hand, the conducted study on flow separation and effects of surface roughness at different rotating speeds has shown that the increments of lift coefficient depends on surface roughness of the cylinder [11, 12]. Moreover, the supercritical flow past a cylinder has been examined, assuming fully turbulent boundary layer [13]. The ranges of the parameters that can affect the rotating cylinder flight have been found to be $1 \geq Power$-Law Index, $n \geq 0.2$, $0.1 \leq Re \leq 40$ and $0 \leq \alpha \leq 6$, but no lifting is produced with positive drag [14].

In summary, the research on rotating cylinder has been proven to be beneficial in many industries including aerospace. However, there is still a research gap in explaining the use of spinning cylinder as a good lift generator. The main objective of this study is to analyze the effect of steady air flow and concentrated airflow in 2D numerical simulation on rotating cylinder as a potential lifting devices for short take-off flying distance aircraft.

2. Methodology

For validation purpose of the numerical analysis, a rotating cylinder model used in the experiment in Ref. [15] is applied in this study for comparison. The ANSYS Fluent software is used to analyze the fluid flow characteristics on the rotating cylinder and the details of the meshing are tabulated in Table 1. The orthogonal quality value is close to 1, which indicates a good quality of meshes. Furthermore, as shown in Table 2, using the statistical analysis software Statistical Package for the Social Sciences (SPSS), the simulated results are indicated to be essentially 90.6% similar to the experimental results in Ref. [15]. This positive agreement between the two results is taken as a validation for the numerical setup and it can be used to simulate the fluid flow characteristics of the rotating cylinder at different $\alpha$.

| Parameter | Value |
|-----------|-------|
| Nodes     | 62704 |
| Elements  | 62345 |
| Min       | 0.590 |
| Max       | 0.999 |
| Average   | 0.986 |
| Standard Deviation | $2.3 \times 10^{-3}$ |

| Parameter | Value |
|-----------|-------|
| Model     | 0.952a |
| R Square  | 0.906 |
| Adjusted R Square | 0.901 |
| Std. Error of the Estimate | 0.331091 |

aPredictors: Rotational speed, air velocity
Dependent variable: Rotational rate

The main interest of this study is to improve the lift coefficient of the cylinder. Therefore, several configurations to obtain high lift coefficient are considered for this study. A wall block of size 2mm x 15mm is located near the cylinder to evaluate and analyze the Magnus effect. For the concentrated air flow, the velocity inlet is placed nearby to minimize turbulent boundary layer that causes drag. For the simulations, the fixed constants are set as follows: $Re = 2.2 \times 10^4$, inlet velocity, $U_\infty = 7$ m/s, $\alpha = 1.54$ and rotational speed, $U_0 = 718.67$ rad/s (clockwise). Figure 1 illustrates the geometry model for steady and uniform air flow whereas Figure 2 is showing the geometry model on air inlet shifted near-front with different inlet opening size in front of rotated cylinder for concentrated air flow simulation.
The mesh quality is important for domain discretization. The skewness is evaluated by FLUENT. Meshes have been generated for both testing where Minimum Orthogonal Quality should be close to 1 [16]. Figure 3 shows the meshing for steady and concentrated air flow approaches, where the meshes for determining optimum $C_L$ satisfied at Minimum Orthogonal Quality of $6.148 \times 10^{-1}$ and optimum $C_D$ at Minimum Orthogonal Quality of $5.809 \times 10^{-1}$. These values are more than 0.5 and close to 1, which can be considered that a good quality of mesh has been generated.

The flow behaviour near the wall is complicated and also hard to distinguish with that at different regions near the wall. Hence the concept of $y^+$ has been formulated, which is a dimensionless quantity that corresponds to the distance from the wall measured in the terms of viscous length. It is related to the turbulence modelling, especially mesh generation processes, and how it is going to affect the flow simulation result [16]. A non-dimensional wall distance for a wall-bounded flow can be determined by using Equation 1, where $u^*$ is the friction velocity at the nearest wall, $y$ is the distance to the nearest wall and $v$ is the local kinematic viscosity of the fluid. $y^+$ is often referred as $y$ plus and is commonly used in boundary layer theory and in defining the law of the wall.

$$y^+ = \frac{(u^*y)}{v}$$ (1)
For external flows, the calculation of wall height starts from the skin friction coefficient, $C_f$, as given by Equation 2 [14].

$$C_f = 0.0508 \text{Re}^{0.2}$$  \hspace{1cm} (2)

The wall shear stress, $\tau_w$, can be calculated from skin friction coefficient as in Equation 3. The value is then used to calculate the frictional velocity, $U_f$, as in Equation 4. Finally, the wall height is estimated by Equation 5.

$$\tau_w = 1/2C_f \rho U^2$$  \hspace{1cm} (3)

$$U_f = \sqrt{\tau_w / \rho}$$  \hspace{1cm} (4)

$$y = \left( y^+ + v \right) / U_f$$  \hspace{1cm} (5)

For the simulation in this study, the fluid is assumed to be an ideal gas having constant specific heats capacity. The flow analyses are carried out using commercial CFD package, ANSYS CFX that executes the 3D RANS equations based on the finite volume numerical method. The solver type used is pressure based with absolute velocity formulation, time considered at transient on 2D space planar. The air flow is set as ‘viscous’ and is reflected to ‘SST K-Omega, K-ω’. The values of air density, $\rho$, of 1.225 kgm$^{-3}$ and viscosity, $\mu$, of 1.789 x 10$^{-5}$ are used. Moreover, for boundary conditions, the cylinder is fixed as a wall but to rotate the cylinder, momentum parameters are clarified for moving wall that is ‘relative to adjacent cell zone’ with ‘rotational speed’. To prevent of any other disturbance, the shear condition is set to ‘no slip condition’ with ‘wall roughness height (m)’ is set to zero. Since the air flow considered is turbulent flow, the specification method is set for ‘intensity’ of 5% and ‘viscosity ratio’ of 10. The same setting is used for the outlet velocity.

The algebraic equations are solved in iterative manner, which results in velocity and pressure fields being updated together. A fully implicit coupling has been achieved by implicit discretization of the pressure gradient terms from the momentum equations, including implicit discretization of the mass flux. All procedures are linked with an algebraic multi-grid method and the set of equations is solved by a point or block using the Gauss-Seidel technique. On the other hand, to solve the equations for the unsteady form, second-order discretization scheme is used to integrate the equations in time domain. This scheme is implicit and the solving process is done with numerical stability. The time step is chosen based on the dimensionless step [16].

### 3. Results and Discussion

The optimization of the coefficient of aerodynamics forces, either $C_L$ or $C_D$, has been carried out with a wall block is located near to the rotated cylinder body to restrict incoming air flow. It is important to see the effect in order to define the optimum gap at Y-axis, $G_Y$ for steady source of air to increase the speed of air at the upper side of the cylinder body. Five configurations are considered in this study for high coefficient of lift while three configurations are considered for low coefficient of drag. Table 3 and Table 4 tabulate the results in a steady air flow for optimum $C_L$ and $C_D$, respectively.

From Table 3, Configuration 3 has shown the highest resultant $C_L=1.725$ with $G_Y=10$ mm, which is actually lower than to the on-design operating point $C_L=2.170$ (no wall block). The simulation result for this Configuration 3 is illustrated in Figure 4. The velocity distribution contour is shown in Figure 4(a), where 10% to 20% of the cylinder perimeter operating points located at the top are considered to be at high velocity while the low velocity occurred at the lower separation region. The vortex wake is identified at half of the cylinder diameter size. Furthermore, Figure 4(b) describes the upper boundary layer streamlines that is thinner than the lower boundary layer. The velocity streamlines path indicates a turbulent state at the back of the cylinder. Meanwhile, Figure 4(c) implies that 10% - 20% of the cylinder perimeter operating points located at the lower separation region are considered high pressure and the low pressure region occurred at the top of the operating cylinder.

In the meantime, from Table 4, Configuration 2 has shown the lowest result for $C_D$ with 0.027 at $G_X=30$ mm. This result is an improvement in comparison to the on-design operating point $C_D=0.298$ (no
wall block). The velocity distribution contour in this case is displayed in Figure 5(a), which shows at least 25% of the cylinder perimeter operating points at the upper separation and secondary boundary layer region are considered high velocity. The vortex wake is greater behind the cylinder but it rapidly decreases. Figure 5(b), on the other hand, highlights that the thinning of the upper separation region streamline has started at the boundary layer origin (stagnation point) where the air velocity is higher than the lower separation region streamlines. Moreover, Figure 5(c) shows that at least 10% - 20% of the cylinder perimeter operating points are considered high pressure at the stagnation point where the boundary layer originated. Low pressure is identified at the upper separation region.

Table 3: $C_L$ optimization in steady air flow

| Configuration | $G_Y$ (mm) | $C_L$  | $C_D$  |
|---------------|------------|--------|--------|
| 1             | 5.0        | 1.048  | 0.615  |
| 2             | 7.5        | 1.296  | 0.607  |
| 3             | 10.0       | 1.725  | 0.488  |
| 4             | 12.5       | 1.118  | 0.485  |
| 5             | 15.0       | 0.953  | 0.481  |

Table 4: $C_D$ optimization in steady air flow

| Configuration | $G_X$ (mm) | $C_L$  | $C_D$  |
|---------------|------------|--------|--------|
| 1             | 45         | 0.507  | 0.055  |
| 2             | 30         | 0.501  | 0.027  |
| 3             | 15         | 0.523  | 0.039  |

Figure 4: Simulation result of Configuration 3 in steady air flow for $C_L$ optimization: (a) velocity contour, (b) streamlines, (c) pressure contour

Figure 5: Simulation result of Configuration 2 in steady air flow for $C_D$ optimization: (a) velocity contour, (b) streamlines, (c) pressure contour

Meanwhile, for the concentrated air flow cases, there are four considered configurations in the $C_L$ optimization and five configurations for $C_D$ optimization. Table 5 shows the detailed simulation results for $C_L$ optimization study while Table 6 tabulates the obtained results in $C_D$ optimization study. From Table 5, the highest $C_L$ of 5.316 is obtained by Configuration 2 with $G_Y=35$mm and $G_X=5$mm. This is higher in comparison to the on-design operating point $C_L$ of 2.170 (normal air inlet distance). Figure 6(a) shows the simulated velocity distribution contour for Configuration 2, where at least 40% - 50% of the upper separation and 10% - 15% of the lower separation cylinder perimeter operating points are considered of high velocity. In addition, Figure 6(b) shows the thinning of the streamlines at the upper separation point. The stagnation or boundary layer origin can be clearly seen to be shifted upward and deflect the air from the upper to lower regions. Figure 6(c) depicts that at least 10% - 20% of cylinders perimeter operating points with the highest pressure are located at lower separation point whereas the
upper separation region is considered lower pressure. As for the optimum configuration for $C_D$, based on the results in Table 6, Configuration 5 appears to have the lowest $C_D$ with 0.122 at $G_Y=1492.5\text{mm}$ and $G_X=10\text{mm}$. This is an improvement compared to the on-design operating point $C_D=0.298$ (normal air inlet distance). The velocity distribution contour for Configuration 5 is displayed in Figure 7(a), which shows that 70% - 80% of cylinder perimeter operating points at the upper separation (which in the primary boundary layer region) is considered have the highest air velocity. Figure 7(b) shows that the upper separation region has gained the highest air velocity and the stagnation point has shifted a little below than the normal condition of Magnus effect. Last but not least, in Figure 7(c) displays that at least 10% – 15% of cylinders perimeter operating points that are considered as having the highest pressure are located at the stagnation point whereas the lowest ones are located at the upper separation region close to the wall domain.

Table 5: $C_L$ optimization in concentrated air flow

| Conf. No. | Inlet size (mm) = 1500 mm - 15 mm + $G_Y$ | $G_X$ (mm) | $C_L$ | $C_D$ |
|-----------|----------------------------------------|-----------|-------|-------|
| 1         | 40.0                                   | 10        | 3.788 | 3.331 |
| 2         | 35.0                                   | 5         | 5.316 | 4.325 |
| 3         | 45.0                                   | 15        | 2.464 | 2.473 |
| 4         | 22.5                                   | 15        | 0.742 | 0.189 |

Table 6: $C_D$ optimization in concentrated air flow

| Conf. No. | Inlet size (mm) = 1500 mm - 15 mm + $G_Y$ | $G_X$ (mm) | $C_L$ | $C_D$ |
|-----------|----------------------------------------|-----------|-------|-------|
| 1         | 1515.0                                 | 5         | 2.818 | 2.522 |
| 2         | 1507.5                                 | 5         | 2.245 | 1.765 |
| 3         | 1500.0                                 | 5         | 1.570 | 1.074 |
| 4         | 1500.0                                 | 10        | 1.869 | 0.754 |
| 5         | 1492.5                                 | 10        | 0.757 | 0.122 |

4. Conclusion
The main objective of this study is to investigate the effects of steady and concentrated air flows on the rotating cylinder. Based on the simulation results for the steady air flow setup, it has been found that optimum configuration for the highest $C_L$ produces a lower value, $C_L=1.725$ in comparison to that
generated without the wall block configuration, $C_L=2.170$. In contrast, the simulated $C_D$ result for the optimum configuration shows big improvement with $C_D=0.027$, which is lower than $C_D=0.298$ for without the block configuration. Meanwhile, for concentrated air flow setup, it is shown that the optimum $C_L$ value of 5.316 is a great improvement compared to the on-design value of $C_L=2.170$. As for the optimum $C_D=0.122$, the result is better than $C_D=0.298$ for the on-design. All in all, it can be concluded that, with the right configuration of the concentrated air flow setup, the rotating cylinder can be used as a lifting device for very short take-off since it can produce very high coefficient of lift (2.5 times higher) compared with steady air flow configuration.

Acknowledgement
The authors like to thank Assoc. Prof. Dr.-Ing. Surjatin Wiriadidjaja for his support with the fundamental aerodynamics knowledge and application in this study. The authors also acknowledge financial support from Universiti Putra Malaysia through research grant GPB 9531600 and GP-IPB9415402.

References
[1] Seifert J 2012 Prog. Aerosp. Sci. 55 17–45
[2] Collis S S, Joslin R D, Seifert A and Theofilis V 2004 Prog. Aerosp. Sci. 40 237–89
[3] Carstensen S, Mandviwalla X, Vita L and Paulsen U S 2014 Journal of Ocean and Wind Energy 1 41–9
[4] Gowree E R and Prince S A 2012 28th Int. Congr. Aeronaut. Sci.
[5] Seifert J 2012 50th AIAA Aerospace Sciences Meeting
[6] Rao A, Stewart B E, Thompson M, Leweke T and Hourigan K 2011 J. Fluids Struct. 27 668–79
[7] Lam K M, Liu P and Hu J C 2010 J. Fluids Struct. 26 703–21
[8] Rostamy N, Sumner D, Bergstrom D J and Bugg J D 2012 J. Fluids Struct. 34 105–22
[9] Harichandan A B and Roy A 2012 J. Fluids Struct. 33 19–43
[10] Boudaoud W, Yahiaoui T, Imine B and Imine O 2012 EPJ Web Conf. 25 1104
[11] Marzuki O F, Mohd Rafie A S, Romli F I and Ahmad K A 2017 Acta Mechanica
[12] Marzuki O F, Mohd Rafie A S, Romli F I and Ahmad K A 2015 ARPN Journal of Engineering and Applied Sciences 10 9725-9
[13] Karabelas S J, Koumroglou B C, Argyropoulos C D and Markatos N C 2012 Appl. Math. Model. 36 379–98
[14] Panda S K and Chhabra R P 2010 J. Nonnewton. Fluid Mech. 165 1442–61
[15] Reid E G 1924 Tests of Rotating Cylinders National Advisory Committee for Aeronautics
[16] ANSYS Inc. 2004 ANSYS Modeling and Meshing Guide