Computational Investigation of Micro Helicopter Near-Wall Effect

D. C. Robinson¹, H. Chung¹ and K. Ryan¹

¹Department of Mechanical and Aerospace Engineering
Monash University, Melbourne, Victoria 3800, Australia

Abstract
A key challenge faced by micro rotorcraft operating in very confined environments is disturbance effects that occur due to aerodynamic interactions between rotors and adjacent structures. This study uses computational fluid dynamics to model a micro rotor \((Re_{tip} = 50,000)\) hovering near to a flat vertical wall. The results show that a wall adjacent to a hovering micro rotor will induce two predominant wake asymmetry phenomena: Asymmetry in wake shape; and asymmetry in blade-tip vortex circulation strength. These wake asymmetry phenomena induce asymmetry in the flow field at the rotor disk which causes the lift force acting at each blade to vary with rotor azimuth angle. This lift asymmetry generates significant disturbance moments acting about the rotor disk’s lateral and longitudinal axes. Additional simulations have also shown that as the attitude of the rotor varies by \(\pm 5^\circ\) the disturbance moments vary linearly. The results presented in this study will serve as a fundamental basis for future real-time estimation of aerodynamic disturbances induced by a vertical wall, which will be used to safely control and guide a micro rotorcraft in confined environments.

Introduction
Micro helicopters are a class of rotary wing aircraft that have developed rapidly over the last 15 years [6]. Owing to their small footprint and Vertical Take-Off and Landing (VTOL) capability, they are particularly well suited for carrying out search and rescue, surveillance and reconnaissance missions within very confined environments. By definition, confined environment operations require micro helicopters to fly in close proximity to structures, such as walls, floors, ceilings, narrow ducts and passageways, etc. This poses a unique challenge, as aerodynamic interactions between rotor wakes and nearby structures can induce undesirable forces and moments at the rotor disk. Because micro helicopters have very small inertia, these forces and moments present a significant disturbance that can adversely affect performance and stability. In order to develop reliable micro helicopter systems, it is vital to understand micro rotor fluid-structure interactions so that these effects within confined environments can be detected and compensated for appropriately. An earlier study by the authors began to address this issue by modelling a micro rotor hovering near a vertical wall [8]. The results showed that the wall induces significant wake asymmetry which produces large disturbance forces and moments at the rotor disk. As an extension of this, the study outlined in this paper considers the effect of small variations in rotor attitude on fluid-wall interactions.

This paper is structured as follows: computational methodology; validation of computational methodology; results; and concluding remarks.

Methodology
Computational Fluid Dynamics (CFD) has been chosen for modelling the flow effects in this study because it gives good physical insight into flow phenomena that is otherwise difficult to obtain using experimental flow visualisation techniques. The rotor geometry is based on a Blade Nano CPX micro helicopter [2] that has been modified for use as an experimental testbed. It has mass, \(m\), of 36g; rotor radius, \(R\), of 103mm; chord length, \(c\), of 16mm; collective pitch, \(\alpha\), of 7.5°; rotor speed, \(\Omega\), of 3,000RPM and a corresponding blade tip Reynolds number, \(Re_{tip}\), of 50,000. The rotor is modelled as a rigid body and the helicopter body and tail rotor have been neglected for simplification. The fluid domain geometry is cylindrical in shape with a height of 30\(R\) and a radius of 30R. An illustration of the domain geometry is provided in figure 1. The global coordinate system has been define with the \(x\)-axis toward the wall, the \(y\)-axis parallel to the wall and the \(z\)-axis aligned with the rotor axis of rotation. The computational grid has been generated using an unstructured tetra Delaunay method [1, 11] with prism layers at the blade surface and increased resolution in the blade tip, blade surface and wake regions. Cross sections of the mesh through the rotor are provided in figure 1. A moving mesh method has been used to model rotor rotation with a temporal discretisation of 1.8° of rotation per simulation timestep. The far-field boundary conditions have been prescribed using a source-sink method [10]. The flow throughout the fluid domain is solved using a second order implicit solver with a Spalart-Allmaras turbulence model [3, 4]. The presence of a wall within the fluid domain is modelled by forcing the flow velocity to zero within an immersed boundary region [5].

Validation of Computational Methodology
A mesh independence study has been conducted to ensure that the results obtained are independent of the mesh resolution. Three simulations with a wall gap of 2\(c\) were run using three different meshes with varying resolutions. Based on a Richardson extrapolation estimate of error, it is estimated that the peak error present in the solution due to mesh dependency remains less than 1% throughout the flow.

Figure 1. Domain geometry and mesh cross sections at rotor
Additionally, a domain size study has been conducted to ensure that the results obtained are independent of the domain size. A simulation was first run using a nominal domain size (domain height of 30R and domain radius of 30R). Following this, two additional simulations were run, one with the domain height doubled to 60R and one with the domain radius doubled to 60R. The results of the domain size study show that peak error (measured relative to the nominal domain) remains below 1%.

Finally, an evaluation of turbulence models has been conducted. The Spalart-Allmaras [9] turbulence model was selected owing to its close agreement with experimentally obtained results and its previous use in the literature [3, 4].

Results
Using the methodology outlined above, CFD simulations have been run with no wall in the fluid domain and with a wall placed between 2c and 6c from the rotor disk (increasing in increments of 2c) and with the blade rotating in a clock-wise direction (when viewed from above) at 3,000RPM. To account for small fluctuations in attitude that a micro helicopter will encounter when near hover, the attitude of the rotor disk, β, has been varied by ±5° from horizontal about the y-axis. The resulting wake distortion and rotor-wall induced forces and moments are analysed in the following sections.

Wake Distortion
A qualitative analysis of the micro rotor near-wall effect is presented in Figure 2 in the form of cross sections of vorticity contours. Figure 2 (a) shows a vorticity contour with no wall present near the rotor. The no-wall case exhibits a radially...
contracting wake that is consistent with low blade-tip Reynolds number rotors at hover from the literature [7, 3]. Figure 2 (a) and (b) show vorticity contours taken with the rotor normal and parallel to the wall respectively. Wake shape asymmetry is least pronounced in the $\gamma$-direction (parallel to the wall), as shown in figure 2 (b). This is because the fluid flow is not inhibited in the plane parallel to the wall and, therefore, radial wake contraction occurs readily in this plane. In contrast, there is clear asymmetry in the wake shape as the blade tip passes the wall in figure 2 (c). This is because fluid has been prevented from entering the rotor disk from the direction normal to the wall; as such, fluid must enter the rotor disk from above (parallel to the wall) and the wake convects downwards with little radial contraction as a result.

**Blade Tip Vortex Position and Circulation**

The wake distortion presented in the preceding section directly affects the position and circulation of the helical blade tip vortex wake. This can be shown by defining blade tip vortices at a vertical plane $20^\circ$ behind the blade. The circulation in the primary and secondary blade tip vortices is calculated by taking the surface integral of vorticity across a circle with radius of $\frac{R}{2}$ and origin at the local maximum vorticity,

$$\Gamma = \int \int_S \omega \cdot dS,$$

where $\Gamma$, $S$ and $\omega$ are circulation, vortex surface and vorticity respectively. A plot of vortex circulation strength for the primary and secondary vortices vs rotor azimuth angle, $\psi$, is provided in figure 3, where circulation is non-dimensionalised by,

$$C_\Gamma(\psi) = \frac{\Gamma(\psi)}{2\pi A \Omega},$$

where, $C_\Gamma$, $\Gamma$, $A$ and $\Omega$ are non-dimensionalised circulation, circulation, rotor disk area and rotor angular speed respectively. Note that the legend provided in figure 3 is consistent with the remaining figures in this paper. The plot shows circulation for a wall $2c$ from the blade tip and with rotor attitude, $\beta$, of $-5^\circ$, $0^\circ$ and $5^\circ$ about the $\gamma$-axis. The blade tip is nearest the wall at $\psi = 0^\circ$ and is farthest from the wall at $\psi = 180^\circ$. The plot shows a significant reduction in circulation between $\psi = 0^\circ$ and $\psi = 90^\circ$ (i.e.; in the quarter rotation after the blade tip has passed the wall). At it’s minimum, the circulation is 85.5% of the no wall at $\psi = 72^\circ$ for the primary vortex and 80.4% of the no wall at $\psi = 72^\circ$ for the secondary vortex. This result can be explained by the presence of the wall inhibiting air flow into the wake between $\psi = 0^\circ$ and $\psi = 90^\circ$. From $\psi = 90^\circ$ and $\psi = 360^\circ$, the circulation is closer to a constant level for both the primary and secondary vortices because air flow into the wake is less inhibited. For variations of $\beta$ between $\pm 5^\circ$ a small, yet still detectable, variation in circulation is present.

Similarly, wake distortion affects the location of blade tip vortices. Plots of radial position and $\gamma$-axis position of the primary and secondary vortices $20^\circ$ behind the blade are provided in figure 4 and figure 5 respectively, where position is non-dimensionalised by rotor radius, $R$. For the primary vortex, position variation is small, varying between 100.1% and 100.9% of the no wall position in the radial direction and 116.6% and 83.3% of the no wall position in the $\gamma$ direction. In contrast, the secondary vortex position variation is larger, varying between 108.1% and 98.8% of the no wall position in the radial direction and 122.2% and 85.4% of the no wall position in the $\gamma$ direction. The reason for this variation can be explained by the wall inhibiting flow in the direction normal to the wall. In order for momentum to be conserved, the flow is forced to be predominantly parallel to the wall in this region.

**Forces and Moments Acting at Rotor Disk**

The variation in circulation and location of the helical blade tip vortex wake directly induces variation in the forces acting at the rotor disk. A plot of lift force acting on a single blade vs rotor azimuth angle is provided in figure 6, where lift force is non-dimensionalised by,
The results of the CFD simulations presented in this study show that a wall near to a hovering micro rotor will induce asymmetry in the circulation and position of the helical blade tip vortex wake. Asymmetry in vortex wake circulation and position directly induces asymmetry in lift forces acting at the rotor disk. For blade near a wall, lift force fluctuates periodically as a function of rotor azimuth angle, with minimum lift occurring at $\psi = 12.6^\circ$ and maximum lift occurs at $\psi = 147.6^\circ$. These fluctuating lift forces induce moments acting at the rotor disk that vary as a function of rotor-wall separation and rotor attitude. As part of proposed future work, the CFD results presented in this paper will serve as a basis for design of micro helicopter state estimators and controllers.

Conclusions

References