Estimating Vertical Drag on Helicopter Fuselage during Hovering

A. A. Wahab[*] and M.Hafiz Ismail[**]
Aeronautical & Automotive Dept., Faculty of Mechanical Engineering,
Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia.
FAX: +607-5566159 **TEL**: +607-5534674
abas@fkm.utm.my

Abstract: This research aims at prediction of aerodynamic drag on a conventional helicopter fuselage, EC-120. The drag has been evaluated using theoretical method which is aircraft elements method [1] and numerical method which is using commercial software Fluent 6.2 [2]. The two results are found to be in good agreement. This result is being confirmed by the drag value of similar group helicopter.

Keywords: Vertical Drag, Aircraft Element Method, Moving Reference Frame

Notation

D_v	: Vertical Drag, N
GW	: Gross Weight, N
DL	: Disc Loading, m^2
$M= v/a$	: Mach Number
$a = 346$	: Speed of Sound, ms^{-1}
$C_T/\sigma = 0.061$	: Average Two Dimensional Lift Coefficient

Introduction

The flow field around a rotor, whether in forward flight or hover, is difficult to model due to the presence of strong vorticity. The effect of flow from rotor will cause drag on the helicopter thus contribute download for the helicopter. The study of rotor fuselage flow interaction is very important since fuselage drag has been shown to account for up to one-third of total helicopter drag. In addition, the rotor should be included in any numerical simulation since rotor-fuselage interaction are complex and may have a major influence on the helicopter flow field physics [3]. In all cases involves drag, the aerodynamic forces and moments on the body are due to only two basic sources, pressure distribution over the body surfaces and shear stress distribution over the body surface. No matter how complex the body shape may be, the aerodynamic forces and moments on the body are due entirely to the above two basic sources [4].

The main rotor of helicopter provides three basic functions, the generation of a vertical lifting force in opposition to the aircraft weight, the generation of a horizontal force propulsive force for forward flight, and a means of generating forces and moments to control the altitude and position of the helicopter [5]. Due to these important roles, wake and vortex system generated by the main rotor-blades and interacts with the body surface in a complex flow. Rotors and airframes designed without taking into effect the rotor-airframe aerodynamic-interaction often tend to perform much below their design capability.

This paper presented to investigate the drag caused by main rotor blades in hovering, and use two methods for obtaining the drag. The first is using aircraft elements method that elaborates in [1] and the second method is using computational method which is simulation using commercial software Fluent 6.2 [2]. These two method is compared each other and the percentage different is shown. The helicopter used in this paper is Eurocopter EC120, which is a 4 seater helicopter. Technical specification of the helicopter is shown in Appendixes A. This helicopter is choosing because it can be categorized as small and light helicopter. Thus, the shape is general compared to high speed helicopters. Moreover, complete set of data for technical specification for this type of helicopter has been obtained from Eurocopter Malaysia Bhd. The data is very useful for helicopter analysis.

---

[*] Professor of Aeronautic, Faculty of Mechanical Engineering, Universiti Teknologi Malaysia.
[**] Master Student.

Aircraft Element Method

A method for making a rough estimate of the vertical drag penalty in hover stated as follows

$$\frac{D_V}{GW} = \frac{(0.3)(DL)(projected\_area)}{(DL)(disc\_area)} \tag{1}$$

By referring table 2 in appendixes, substituting in the equation, a first estimate of vertical drag during hovering is

$$D_V = \frac{0.3 \times 8.1}{78.55} \times 13243.5 = 409.7N$$

To get a better result and raise the confidence level in the hover performance calculations, the following step is done for EC120 fuselage body.

i. Divide the plan view of the airframe into segments.
ii. Estimate the drag coefficient of each segment as a function of its shape.
iii. Determine the distribution of dynamic pressure in the rotor wake.
iv. Sum the effect of its segments.
v. Calculate the rotor thrust as the sum of the weight and the vertical drag.

The drag coefficient for each segment depends on the shape of its cross section. Drag measurement of cylinders and flat plates under a rotor are reported in Figure. Using the drag coefficients and the dynamic pressure corresponding to each of the airframe segments, the vertical drag penalty can now be calculated by summing the product of the drag coefficient, the dynamic pressure ratio, and the projected area of each segment in the plan view [1].

$$\frac{D_V}{GW} = \frac{2\sum_{n=1}^{N} C_{d_n} \left({}^{q}\!/_{DL} \right)_n A_n}{A} \tag{2}$$

$$\frac{D_V}{GW} = \frac{2(1.2521)}{78.55} = 0.0319$$

$$D_V = 0.0319 \times 13243.5 = 422.21N$$

Computational Method

Computational simulation of such complicated flows becomes increasingly important in attempts to minimize costly wind tunnel experiments. Historically, rotorcraft simulations employed rotor disk models that were coupled with 3D Navier-Stokes or Euler solvers. To achieve this objective, commercial software Fluent 6.2 is used to estimate vertical drag during hovering on the desired helicopter fuselage.

Fluent 6.2 provides comprehensive modeling capabilities for a wide range of incompressible and compressible, laminar and turbulent fluid flow problems. To permit modeling of fluid flow and related transport phenomena in industrial equipment and processes, various useful features are provided. These include porous media, lumped parameter (fan and heat exchanger), streamwise-periodic flow and heat transfer, swirl, and moving reference frame models. Flows from main rotor blades that contribute vertical drag on helicopter fuselage are categorized as rotating flow. For rotating flow model, Fluent 6.2 provides Fan Model, Mixing Plane Model, Sliding Mesh Model, Dynamic Mesh Model and Moving Reference Frame. The Moving Reference Frame (MRF) family of models includes the ability to model single or multiple reference frames [2]. MRF is choosing as a solution approach because of its reliability among others.

When such problems are defined in a rotating reference frame, the rotating boundaries become stationary relative to the rotating frame, since they are moving at the same speed as the reference frame. For example, the blades rotates clock wise and induced flow through disc thus create lift. By modeling using MRF, the fluid adjacent to the blades is moving, but in opposite direction compare to the blades.

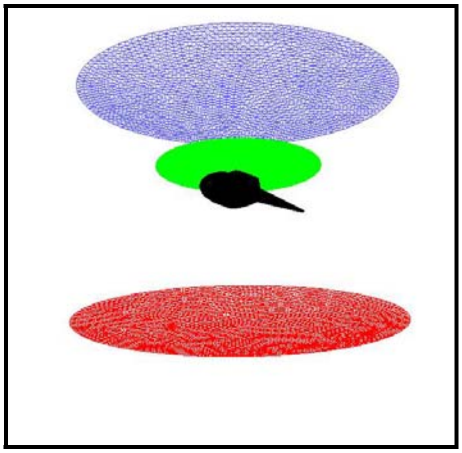

Fig. 1: Modeling Using Fluent 6.2

Figure 1 shows helicopter fuselage with main rotor blades in a domain covers by pressure inlet boundary condition (upper side) and pressure outlet boundary condition (lower side).

Turbulence model is used in the simulation because the main rotor blades rotate at high velocity and reach M=0.66, correspond to Reynolds number of 36.5×10^5. Flow is fully turbulent at Reynolds number more than 30×10^5 [6]. For the wall-bounded flows that were of most interest when the model was formulated, turbulence is found only where vorticity is generated near walls. The best turbulence model to suit this problem is by using Spalart-Allmaras turbulence model that available in Fluent 6.2. The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematics eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients [2].

The simulation is then carried out until mass flow rate is stable through inlet and outlet. More flow visualization is shown in appendixes. For this simulation, 8 models are used. These models vary from the location of main rotor blades position in azimuth angle, ψ which is defined zero when blade is pointing downstream. In hovering flight, the velocity variation along the blade is azimuthally axisymmetric and radially linear, with zero flow velocity at the hub and reaching maximium velocity at the tip [5]. Figure in appendixe shows models and main rotor blade position, and table shows azimuth angle for each main rotor blades. The simulation is carried for various main rotor blades azimuth position because of these assumptions, there is no overlapping flow among the models and all models contains main rotor blades covering disc area so that airflow fully generated by rotor disc. Rotor wake visualization can be seen in figure in appendixe.

Result and Discussion

Final result of hovering drag on fuselage is shown in the following table. First estimation of the vertical drag during hovering is by determining the projected area of fuselage planview and assuming an effective drag coefficient of 0.3 for all aircraft components in the remote wake. This assumption is proven to be useful for first estimation of vertical drag during hovering for most helicopters [1]. Then to raise the level of confidence, aircraft element method is used. Table in appendixes shows detail section with appropriate values.

Meanwhile, the value obtained from simulation varies among models. Because of the variation, the data obtained are plotted and shown in Fig. 2. It can be seen from the graph, vertical drag on the fuselage during hovering reach the maximum drag at model 1 where the value is 409.8 N, and minimum vertical drag occur in model 3 where the value is 152.1N. During hovering, vertical drag on the fuselage varies; depend on the location of main rotor blades. Model 1 shows maximum vertical drag, whereas a blade locate at $\psi^0=0$. It can be seen from fig. that the blade is proportional to the fuselage body. Thus, most induced velocity is projected directly to the fuselage surface. In model 3 where minimum vertical drag occurs, all blades locations are far from fuselage surface. This position explains why minimum value obtained.

Table 1: Vertical Drag on the Fuselage during Hovering

Method	**Vertical drag on fuselage, N**	**Percentage different from theory, %**
First estimation	**409.9**	**22.6**
Aircraft element method	**422.21**	**20.3**
Numerical method (Fluent 6.2)	**409.8**	**22.6**
Theoretical value from [1]	**529.74**	**0**

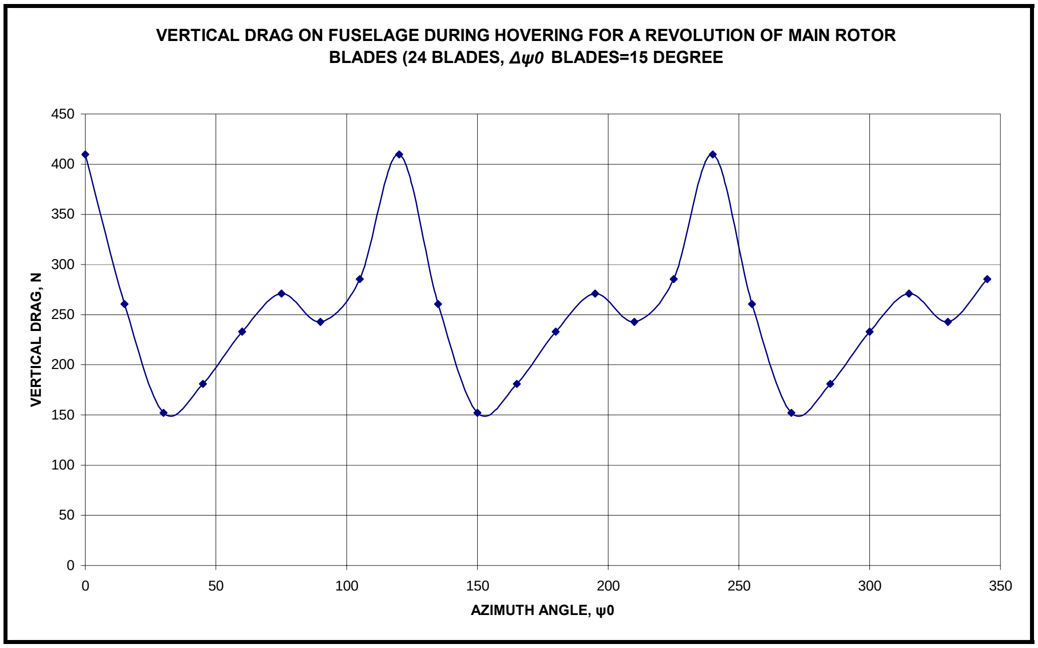

Fig. 2: Variation of Vertical Drag on Fuselage during Hovering For a Revolution of Main Rotor Blades

Conclusion

It can be concluded that the results obtained both by aircraft element method and numerical method compares well. The percentage different is very small due to some errors in simulation. In simulation, the flow is assumed fully turbulent in the domain. In actual cases, the transition flow occurs in most part especially induced velocity generated by main rotor blades.

By referring [1], page 284, using $C_T/\sigma=0.061$, the corresponding D_V/T is 0.04. This value is not much compared with aircraft element method, whereas the value $D_V/T = 0.032$. Moreover, CFD is proven to be powerful tool in predicting vertical drag over fuselage. Rather than giving the results, CFD able to locate the position of blades where the maximum vertical drags occur.

Reference

1. Raymond, W. Prouty, 1986. Helicopter performance, stability and control, *PWS Publishers*
2. FLUENT 6.2 User's Guide, 2005. *Fluent Inc.*
3. Thomas, R., David, O. and Marilyn, S., 2004. Evaluation Of Isolated Fuselage And Rotor-Fuselage Interaction Using CFD, *Proc. AHS 60th'2004*, Baltimore
4. John, D. Anderson, 1984. Fundamentals of Aerodynamics, *Mc Graw Hill Inc.*
5. Gordon, J. Leishman, 2000. Principles of Helicopter Aerodynamics, *Cambridge University Press*
6. Yunus A. Cengel, John M. Cimbala, 2006. Fluid Mechanics – Fundamentals and Applications*, Mc Graw Hill Inc.*

Appendixes

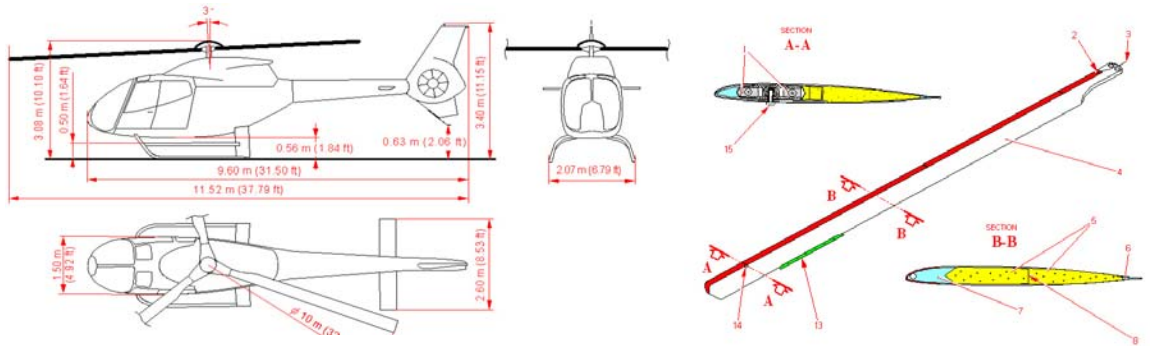

Figure 3: Dimensions of EUROCOPTER COLIBRI 120B and Main Rotor Blades

Table 2: Technical Characteristics and Specifications for EC120B

Gross Weight (GW)	13243.5 N
Rotor diameter	10 m
Number of blades	3
Rotation speed	390 r/min (rpm) to 415 r/min (rpm)
Blade Length	4555 mm
Blade twist	-12^0
Blade Chord	260 mm
Blade Airfoil	NACA 0012
Disc area (A)	78.55 m^2
Fuselage planview area	8.1 m^2
Rotor solidity ratio, σ	0.0496

Table 2: Data Obtained from Aircraft Element Method for EC 120

Airframe segment	Radial position (r/R)	Vertical position (Z/R)	Dynamic pressure ratio (q/DL)	Drag coefficient (C_D)	Segment area (m^2)	(q/DL)x(C_D)x(m^2)
1	0.6	0.3	1.0	0.1	0.24	0.024
2	0.5	0.25	0.8	0.8	0.35	0.224
3	0.4	0.2	0.6	0.8	0.35	0.168
4	0.3	0.18	0.4	0.8	0.36	0.115
5	0.2	0.1	0.25	0.8	0.41	0.082
6	0.2	0.1	0.25	0.8	0.41	0.082
7	0.3	0.1	0.4	0.8	0.335	0.107
8	0.4	0.15	0.5	0.6	0.28	0.084
9	0.5	0.2	0.8	0.6	0.24	0.115
10	0.6	0.25	0.9	0.55	0.21	0.104
11	0.7	0.25	1.0	0.5	0.185	0.093
12	0.8	0.25	0.75	0.45	0.16	0.054
13	0.9	0.25	0	0.4	0.11	0
				TOTAL	3.53	1.2521

Table 3: Main Rotor Blade location and vertical drag on fuselage $\Delta\psi^0 = 15^0$

Main Rotor Blade Azimuth Location	Blade 1 Azimuth Angle, ψ^0	Blade 2 Azimuth Angle, ψ^0	Blade 3 Azimuth Angle, Ψ^0	Vertical Drag On Fuselage, N
1	0	120	240	-409.8
2	15	135	255	-260.5
3	30	150	270	-152.1
4	45	165	285	-181.1
5	60	180	300	-233.1
6	75	195	315	-271.1
7	90	210	330	-242.8
8	105	225	345	-285.3

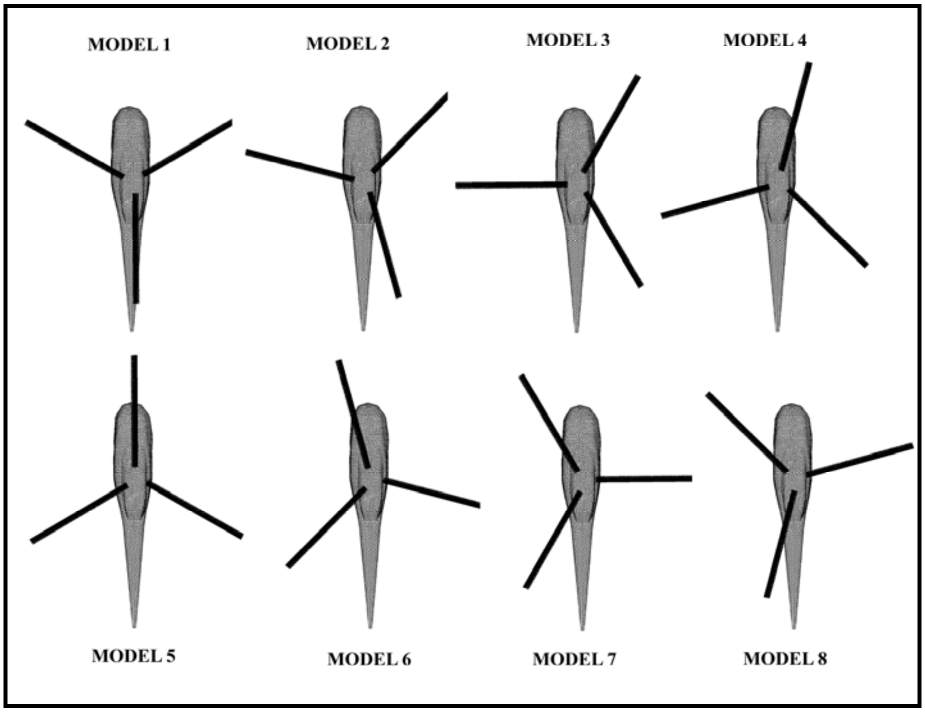

Figure 4: Models for Simulation and Main Rotor Blades Position from Planview

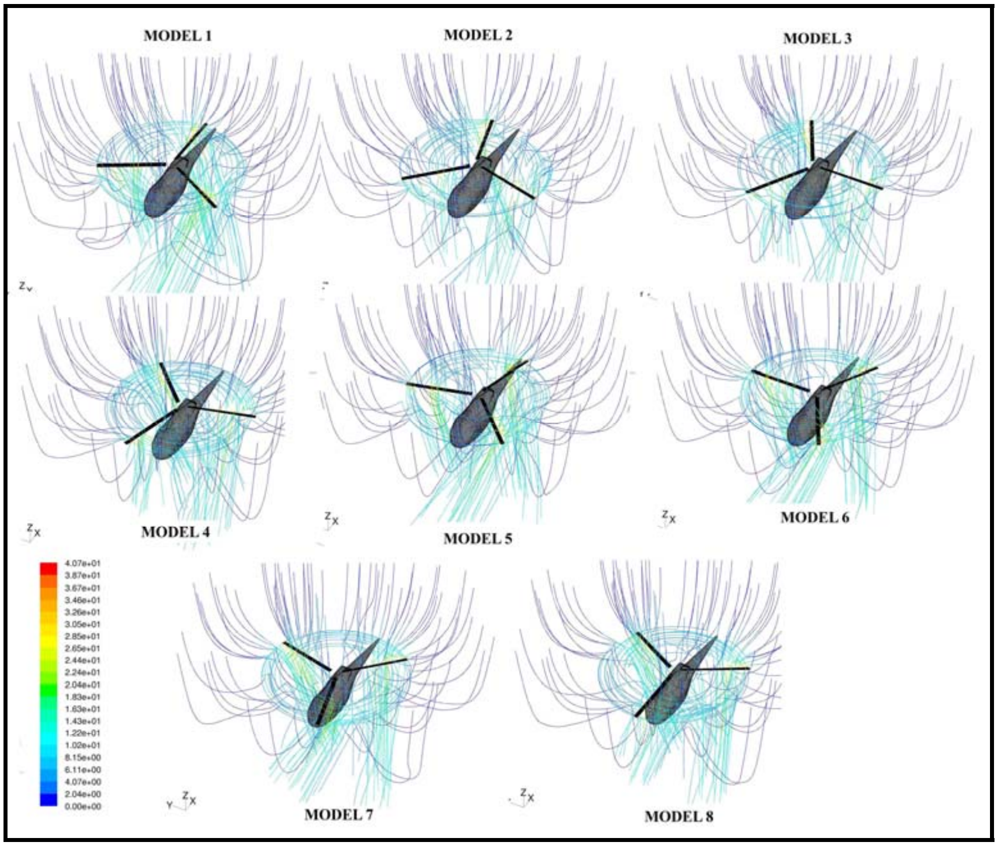

Figure 5: Pathline of Flow from Inlet Colored by Velocity Magnitude

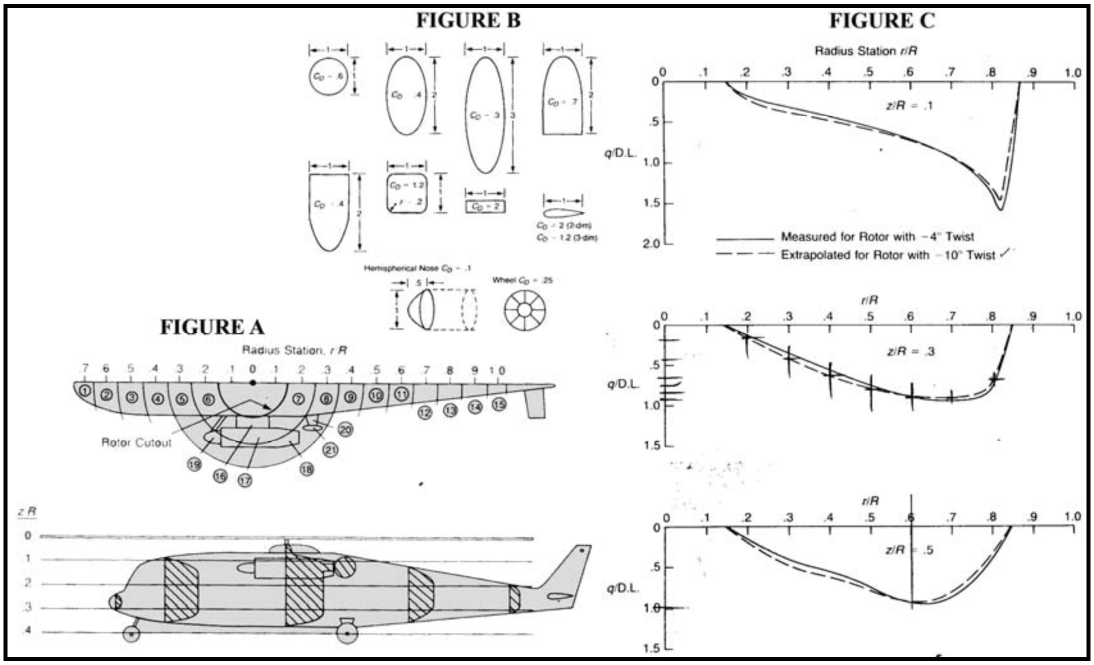

Figure 6: Aircraft Element Method from Prouty