ACOUSTIC ANALYSIS OF ELECTRIC DUCTED FANS

Nick Cunnington-Bourbonniere *[1] **and Joana Rocha** †[1]
[1]Department of Mechanical and Aerospace Engineering, Carleton University, Ottawa, Ontario

1 Introduction

The scope of this paper is to study the initial conceptual design and acoustic analysis of a ducted fan. The fan design was reverse engineered from a reference geometry as used in an UAV (Unmaned Aerial Vehicle) for VTOL (Vertical Take-Off and Landing) applications [1], here called Manta UAV. Ansys CFX was used to analyze the performance [2]. The objective of the analysis was to obtain fan noise, while predicting thrust and torque, with focus in hover flight rather than cruise flight. Electric ducted fans operate with increased efficiency in cruise and reduced operating noise relative to open-rotor configurations. Additionally, the duct can add up to 30% extra thrust in cruise due to the reduction in area allowing for lower power requirement in cruise flight [1]. The smaller fan diameter, however, is less efficient in hover flight.

2 Method

The fan noise was investigated using Ansys CFX. While some of the methods are limited, as they do not consider the duct effects or the rotor-stator interaction, they can be useful to compare different configurations to reduce fan noise in future design iterations. Ansys considers three ways of deriving acoustic energy from kinetic energy [3]. The monopole source is the most efficient technique, the method calculates acoustic energy by forcing gas within the fixed region of space to fluctuate with no reflections at boundaries. Dipole source is predominant in fans during low-speed operation., the method uses two sources that create pulsating spheres that interfere with one another. Lastly, the quadrupole source is least efficient and is best applied when sound is generated aerodynamically without motion of solid boundaries. The current analysis considered that the fluid is compressible, inviscid, without a mean flow, and mean pressure and density are uniform throughout the fluid. The most common way to quantify the acoustic intensity in ANSYS is by using sound pressure level or sound power level. Sound power level was found to be dependant on local reference density as determined by Boussinesq model for buoyant flows [2]. Since this value cannot be calculated analytically, the focus of this analysis was shifted to the sound pressure level (SPL), which is defined by :

$$SPL = 10log_{10}(\frac{P_{ac}}{P_{ref}})^2 \tag{1}$$

with the acoustic pressure defined as following :

$$P_{ac} = \frac{Hn\ Nb^2\ \omega}{2\pi\sqrt{2}d_{obs}cos(\theta_{obs})F_{blade,x}(Hn\,Nb)^{Lc}besselJ(0, Df)} \tag{2}$$

in which Hn is harmonic number, Nb is the number of rotor blades, ω is the absolute angular velocity, Df is the Doppler factor, Lc is the blade loading coefficient, $F_{blade,x}$ is the axial blade loading, and the acoustic reference pressure is $P_{ref} = 2\times10^{-5}$ Pa. A standard sea level speed of sound of $340ms^{-1}$ and a loading coefficient of 2.2 were used. The blade loading coefficient for axial compressors is defined as the ratio of the work done by the blade to the blade velocity squared. The acoustic pressure is derived from the dipole source model as the noise is primarily generated by blade forces. Comparisons were completed for the 1st harmonic frequency.

3 Results and discussion

3.1 Noise versus number of blades per fan

Figure 1 displays the SPL determined at different numbers of rotor blades (Nb) per fan for propulsion configurations of 4, 8, 16, 24 and 36 fans, N_{fans}. While both Nb and $F_{blade,x}$ are variables in Eq. 2, Nb was calculated based on the total thrust, $T = 9.81W$ (in which the total weight is $W = 3000$ kg), divided by the total number of blades of the aircraft, i.e :

$$F_{blade,x} = \frac{9.81\ W}{Nb\ N_{fans}} \tag{3}$$

The results show a noise reduction for configurations with more fans and an increased number of rotor blades per fan ; hence, the selection converged to the 24 fans configuration. While increasing the configuration to 36 fans would decrease hover efficiency, fan noise could be reduced by increasing the number of rotor blades to better distribute the blade loading. A further increase of number of rotors from 31 to 40 would result on a noise reduction of 3 dB.

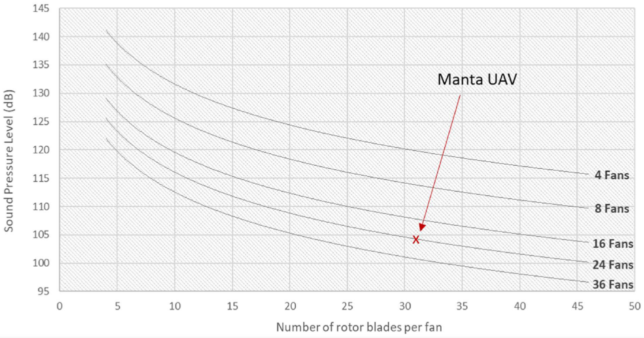

FIGURE 1 – Propulsion configuration vs. noise sensitivity study at varying numbers of rotor blades per fan for different number of ducted fans on aircraft (observer distance of 10 m).

3.2 CFD analysis of the ducted fan

A CFD simulation of the fluid flow within the ducted fan was performed to determine thrust and noise of the Manta UAV

---

*. NickCunningtonBourbo@cmail.carleton.ca
†. Joana.Rocha@carleton.ca

configuration in hover. The rotor was set to rotate at 10000 rpm, a rotational speed towards the higher end of the chosen motor's capability. The rotor shroud was set as a nonslip counter rotating wall with no tip clearance, and an upwind differencing scheme was used. The mixing plane method was chosen to model the interface between the rotor and stator. The inlet-outlet boundary conditions were set as 102 kPa total pressure at the inlet and 101.35 kPa static pressure at the outlet. The turbulence model selected was the shear stress transport model, which combines the k-epsilon model (better away from wall) and the k-omega model (better at wall) to provide turbulence modelling over the entire domain. Two approaches were used to validate the CFD results. First, the analytical prediction (Eq. 1) was compared to the CFD results. Second, a rotor with well documented performance parameters [4] was run with the same conditions applied. The analytical model resulted in a SPL of 108.1 dB for the 1^{st} harmonic, while CFD predicted a SPL of 107.7 dB for the same harmonic. The difference can likely be attributed to the different blade loading, since the ducted fan design is not guaranteed to provide the full/ideal thrust as the one analytical determined (originally considered to be 2452.5 N per fan in hover flight, with a safety factor of 1.5). The same CFD approach and methods used on Manta UAV were applied to the NASA rotor 67 [4]. Table 1 displays the CFD results along with the true values, showing that numerical results agree with the performance data, validating the methods used in the simulation. Figure 2 displays the velocity streamlines for the Manta ducted fan CFD simulation. The flow characteristics resembles what is expected and observed in the rotor 67 (i.e., highest velocity at blade tip), even though the geometries are different. The thrust was determined from the mean-line results, derived from a simple mass balance as following :

TABLE 1 – Comparison of ANSYS CFX results with NASA performance data for the rotor 67 geometry [4].

Parameter	CFD results	NASA data
Mass Flow	33.63 kg s^{-1}	33.25 kg s^{-1}
Total Pressure Ratio	1.65	1.63
Inlet tip Mach number	1.44	1.38

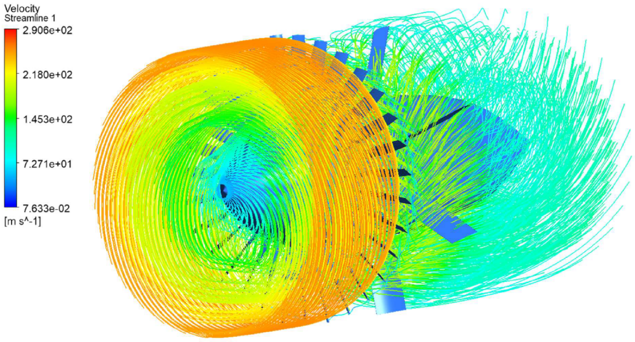

FIGURE 2 – Velocity streamlines for Manta ducted fan geometry.

$$T = \dot{m}_{in}(V_{exit} - V_{in}) + A_{exit}(P_{exit} - P_{in}) \qquad (4)$$

Thrust was determined to be 1990 N per fan from velocity, pressure, and mass flow values obtained from the Manta UAV ducted fan CFD results. The simulation predicted a torque on the rotor of 282.4 Nm. The CFD results show that the used geometry produces less thrust than the one originally considered. This difference could likely be attributed to the degree of flow separation occurring at the rotor (as shown in Figure 3), which could be improved or corrected by increasing the stagger angle of the rotor.

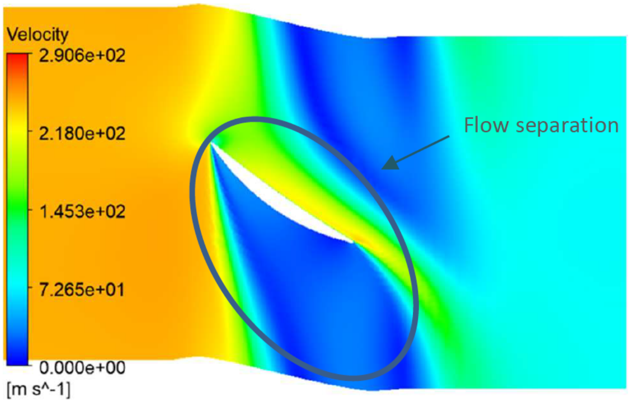

FIGURE 3 – Velocity contour at 0.8 span for the Manta ducted fan.

4 Conclusions

A reversed engineered fan was investigated in order to predict noise, thrust and torque during hover. A sensitivity study was conducted to determine fan noise at different fan configurations. The analysis resulted in the selection of a 24-fan configuration for the Manta UAV, with 31 rotor blades per fan. Analytical predictions estimated a SPL of 108.1 dB for the 1^{st} harmonic, while CFD predicted a SPL of 107.7 dB, validating the CFD model. In addition, the same CFD approach and methods used on Manta were applied to NASA rotor 67, showing that numerical results agree with the performance data. CFD results showed that the rotor-stator geometry produced less thrust compared to initially values used for the design process. This difference could be a result of the degree of separation occurring on the rotor, which could potentially be corrected by increasing the stagger angle of the rotor.

Acknowledgments

The first author would like to thank Prof. Joana Rocha for her guidance during this research.

References

[1] B. Raeisi. Aerodynamic study of tilting asymmetrical ducted fans mounted at the wing tips of a vtol uav. *Thesis, Ryerson University*, 2016.

[2] Ansys cfx-solver theory guide. *ANSYS, Inc.*, 2009.

[3] A. Naseri, M. Boroomand, and S. Sammak. Numerical investigation of effect of inlet swirl and total-pressure distortion on performance and stability of an axial transonic compressor. *Journal of Thermal Science*, 25(6) :501–510, 2016.

[4] A. Strazisar, J. Wood, M. Hathaway, and K. Suder. Laser anemometer measurements in a transonic axial-flow fan rotor. *NASA TP-2879*, 1989.