Computational Analysis of the Blade Number Effect on the Performance of a Ducted Propeller

Carlos M. Lazaro Echavarria¹ and Svetlana V. Poroseva² The University of New Mexico, Albuquerque, NM, 87131

Ducted propellers were found to provide higher propulsive efficiency and operational safety than unducted propellers. The goal of our study is to optimize a ducted propeller design for propulsion of small rotorcrafts. In the paper, the effect of a number of blades on the flow structure around a ducted propeller and in its wake is analyzed by conducting flow simulations. Results are reported for propellers containing from two to five blades. Propeller designs with and without duct are compared. Simulations were conducted using Menter's Shear Stress Transport version of the k- ω turbulence model implemented in the commercial CFD software Star-CCM+. The sensitivity analysis of simulation results to the grid resolution and the time step of simulations is presented.

Nomenclature

= chord of the blades С C_{d0} = section zero-lift drag coefficient C_T = thrust coefficient D = drag FM = figure of merit = lift L = time step Δt = rotor solidity σ = induced power factor κ

I. Introduction

NE of the biggest dreams of humanity is the invention of a "flying car". The developer of the first automobile affordable to middle-class Americans, Henry Ford, once said "a combination airplane and motorcar is coming. You may smile, but it will come." Ford said these words in 1940, but after more than 70 years, the "flying car" idea still remains a dream. The aeronautics engineer, Paul Moller, for example, has tried for more than forty years to make this dream a reality with designs like the Skycar or the Autovolantor¹, but none of them became a success story. However, with improving technologies and, particularly, with computational tools becoming more available for analyzing the performance of various vehicle designs, a perspective of developing a "flying car" may become a reality in the close future.

The goal of our research is to optimize a design of a ducted propeller for the "flying car" propulsion. Previously, ducted or shrouded propellers (also ducted fans) were found to provide higher propulsive efficiency and operational safety². The duct reduces the thrust losses due to the tip leakage flow and generates additional thrust (Fig. 1). In an efficiently designed ducted fan, the majority of thrust can be produced on the shroud. Various companies such as, for example, Trek Aerospace, logiAerospace, and Fuvex employed and analyzed the perceptiveness of ducted propellers in their designs for Vertical Take-Off Landing vehicles³⁻⁵. Shrouded propellers are also widely used in UAV (Unmanned Aerial Vehicle) applications.

¹ Graduate Student, Mechanical Engineering, 1 University of New Mexico MSC01 1150, Albuquerque, NM 87131; also at Escuela Tecnica Superior de Ingenieros Aeronauticos, Pza. de Cardenal Cisneros, 3 28040 Madrid, Spain. AIAA Student Member.

² Associate Professor, Mechanical Engineering, 1 University of New Mexico MSC01 1150, Albuquerque, NM 87131, AIAA Associate Fellow.

There are multiple factors that influence the propeller performance. The objective of our current study is to determine the effect of a number of blades in a given ducted propeller design on the flow structure around the propeller in hover and in its near wake. An initial design is a propeller with untwisted blades attached to the hub at a specified angle. The duct and the blades have the same profile: NACA 64(4)-221⁶. The profile is uniform along the blade. The airfoil was chosen following the general requirements for a rotorcraft airfoil provided by Leishman⁷: high $C_{\rm Lmax}$, high drag divergence Mach number, good lift-to-drag ratio, and low pitching moment.



Figure 1. Schematic diagram of ducted propeller showing thrust generation.

The flow structure analysis was conducted by simulations using

commercial Computational Fluid Dynamic (CFD) software STAR-CCM+⁸. The propeller was designed using the CAD software CATIA⁹. A programming procedure was developed to facilitate generating CAD models for different propeller parameters.

II. Computational ApproachGeneral Guidelines

A. CAD Models

The dimensions of a 5-blade propeller are shown in Fig. 2. The dimensions are based on the blade chord, c = 12.39 cm. The duct diameter is 0.892 m. The duct length is four times larger than the blade chord. The blades are fixed at the constant angle of attack of 10^{0} with no twist and the duct airfoil has no angle of attack.



Figure 2. The 5-blades propeller dimensions (c = 0.1239m).

Simulations were conducted for five propeller designs with two to five blades. Examples of some designs are shown in Fig. 3.



a) Three-blade configuration



b) Five-blade configuration

Figure 3. CAD designs of a ducted propeller.

American Institute of Aeronautics and Astronautics

B. Computational Domain

The computational domain is shown in Fig. 4. In order to incorporate the flow rotation into simulations, a cylindrical control volume subdomain around the propeller was created within the main domain (Fig. 5). The radius of this cylindrical subdomain is 5c and its length is 8c. This control volume was used as a grid refiner as well.



Figure 4. Dimensions of the computational domain.

C. Grid

In the current study, to preserve the grid similarity for different propeller designs and to facilitate the grid generation, a JAVA script was developed. This script also allows generating the grid in parallel.

Structured and unstructured grids were generated using Star-CCM⁸. The comparison of results obtained with two types of grids demonstrated the advantages of the structured grid for the purposes of our research. The structured grids were obtained using hexahedral-like cells (cubes) (Fig. 5). A grid adaptation was used in order to accurately capture flow features. Volumetric controls were used to refine the grid where necessary to resolve the complex flow structure. Figure 5a shows a close view of the grid around a propeller within a cutting plane y=0. Figure 5b shows a general view of the trimmed grid. The rotor rotates in the plane (*x*,*z*) and *y* is the axis of the design.



Figure 5. The Grid around a propeller: two-million-cells configuration.

The ability of many Reynolds-Averaged Navier-Stokes (RANS) CFD methods to predict the wake is usually limited by the numerical dissipation. One way to solve this problem is by using very fine grids. However, this leads to an increase in the required computational time. For flow areas, where the important wake structures such as tip vortices, for example, can be localized, the adaptive mesh refinement (AMR) can be effectively utilized to refine the mesh locally. The application of this technique to simulations of wakes of hovering rotors has been extensively

studied in Refs. 10-12. AMR is also used in our study to resolve the flow features without making computations too expensive. Locations within a flow that require higher resolution (e.g., areas of high pressure and high vorticity) are determined based on the solution-based quantities.

Figure 6 illustrates the application of this technique when the vorticity magnitude is used to determine flow areas for refining. In the figure, the mesh is shown in the plane two chords downstream the blade. In Figure 6a, the mesh is equally spaced. The tip vortex can be seen, but it is weak and its structure is unclear. After a certain number of iterations, an algorithm locates the regions with higher vorticity values and refines them. The result of the grid refining is shown in Fig. 6b. The vortex structure is clearly seen in the figure. Figure 6c shows the result of the second refinement to optimize the mesh.

D. Boundary Conditions

The flow inlet, upstream of the propeller, was set as a velocity inlet with a velocity magnitude of 0 m/s in order to simulate hovering conditions. Additional inlet velocities were also used in simulations: 1 and 2 m/s. The other boundaries of the computational domain were set as pressure outlets with no pressure gradient been imposed on the flow. The propeller surfaces were treated as solid walls.

E. Numerical Methods

The simulations were performed using the CD-adapco CFD software Star-CCM+⁸. Computations were conducted with Menter's



Figure 6. Example of the Adaptive Mesh Refinement application.

Shear Stress Transport (SST) version of the k- ω turbulence model¹⁰. The implicit unsteady model was used with the first-order temporal discretization. The selected flow equation model was the segregated flow. The second-order discretization scheme was used with the implicit integration using the algebraic multigrid (AMG) linear solver for all flow variables. The AMG solver employed a Gauss-Seidel relaxation scheme with a bi-conjugate gradient stabilized acceleration method. A hybrid Gauss-Least Squares Quality (LSQ) method with the Venkatakrishnan limiter was used for the gradients. The second-order convection scheme with the AMG linear solver was applied to the turbulence model equations. A Gauss-Seidel relaxation scheme was also used for the turbulent variables without the acceleration method.

III. Results

A. Grid Sensitivity Analysis

A grid sensitivity analysis was carried out for a five-blade ducted propeller, with the rotational speed being of 6 rps. Three grids with 1.5, 2, and 3 million cells were used in the analysis. The velocity field after 20 seconds of simulation is shown in Fig.7 for the three grids. The figure demonstrates similarity of the velocity fields obtained on grids of 2 and 3 million cells. The same conclusion can be made for the turbulent kinetic energy results (Fig. 8). Some difference in the flow structure obtained with the grids of 2 and 3 million cells is observed upstream of the rotor plane. Currently, we are analyzing possible causes of this effect. Velocity profiles at distances 2c, 5c, 7c and 10c downstream the rotation plane are shown in Fig. 9. The difference between the results of simulations conducted with different grids increases with the distance from the propeller. However, for estimating the propeller performance, the near wake is of more importance. Here, grids of 2 and 3 million cells produce close results.

In Figure 10, the lift coefficient is plotted for the three grids. Its value oscillates between 0.49 and 0.491. For the grids of two and three million cells, the lift coefficient value converged after 10 seconds to the close values. Convergence has not been reached for the grid of 1.5 million cells during this time.

Based on the results of the sensitivity analysis, the grid of two million cells was selected for further simulations.



Figure 7. Grid sensitivity analysis: comparison of velocity fields obtained with the grids of a) 1.5×10^6 , b) 2×10^6 , and c) 3×10^6 cells.



Figure 8. Grid sensitivity analysis: comparison of the turbulent kinetic energy results obtained with the grids of a) 1.5×10^6 , b) 2×10^6 , and c) 3×10^6 cells.



Figure 9. Grid sensitivity analysis: velocity magnitude profiles at different distances downstream the rotational plane.



Figure 10. Grid sensitivity analysis: lift coefficient for the different grids.

6 American Institute of Aeronautics and Astronautics

B. Time-Step Sensitivity Analysis

A sensitivity analysis of simulation results to the time step used in computations was performed for the fiveblade ducted propeller with the grid of two million cells. Simulations with the time step of $\Delta t = 0.1$ s, 0.01s, and 0.001s were conducted. In Figure 11, velocity fields obtained at $\Delta t = 0.1$ s and 0.01 s are compared. Simulations for these time steps were conducted for 40 seconds. As seen in the figure, the wake structure differs considerably in these two cases.

In Figure 12, result of simulations obtained at the three time steps are compared. Due to the computational cost of simulations with $\Delta t = 0.001$ s, preliminary results are available for the total time of four seconds only. The current conclusion is that the wake structures obtained with $\Delta t = 0.01$ s and $\Delta t = 0.001$ s are similar. This is further confirmed by comparing the velocity profiles obtained at different time steps at the distances of 2*c*, 5*c*, and 10*c* below the rotational plane (Fig. 12) and by comparing the lift coefficients C_L (Fig. 13).

At the distance of 2c (Fig. 12a), the velocity profiles are similar for the three time steps. The largest discrepancy between results obtained at different time steps is observed at the rotation axis. Similar observations can be made when comparing the profiles at the distance of 5c (Fig. 12b). At the distance of 10c below the rotational plane (Fig. 12c), the velocity profile obtained at $\Delta t = 0.1$ s differs dramatically from those obtained at smaller time steps.

In Figure 13a, the lift coefficients obtained with the three time steps are compared. The difference between the lift coefficient obtained at $\Delta t = 0.1$ s and at two smaller time steps is clear. However, as shown in Fig. 13b, the difference disappears after a few seconds.

Based on the presented results, the time step of $\Delta t = 0.01$ s was chosen for simulating a flow around the propellers with different numbers of blades.



Figure 11. The velocity field for the ducted propeller with 5 blades for the first 40 seconds obtained with different time steps: a)-d) $\Delta t=0.1s$ and e)-h) $\Delta t=0.01s$.



Figure 12. The velocity magnitude profile for the 5-blade ducted propeller obtained at different time steps at different distances below the rotor plane: a) 2c, b) 5c, and c) 10c.



Figure 13. Lift coefficient for the 5-blade ducted propeller obtained at different time steps at a) t = 4 s, and b) t = 40 s.

C. Analysis of the Blade Number Effect.

The effect of the number of blades on the flow structure around a propeller and in its wake was analyzed for ducted and unducted propellers. Table 1 shows all considered combinations of the simulations parameters: the number of blades, the inlet velocity, and the rotational speed.

Simulation	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24
Blades	2	2	2	2	2	2	3	3	3	3	3	3	4	4	4	4	4	4	5	5	5	5	5	5
m Velocity(m/s)	1	1	1	2	2	2	1	1	1	2	2	2	1	1	1	2	2	2	1	1	1	2	2	2
N(rps)	30	40	50	30	40	50	30	40	50	30	40	50	30	40	50	30	40	50	30	40	50	30	40	50

Figure 14 shows the contribution per the propeller part to the thrust coefficient for the ducted propellers. The average contribution of the blades and the duct were calculated as well. The duct on average contributes up to ~ 15 % to the overall thrust (Fig. 15), but its contribution can reach 30% in some cases.



Figure 14. The average contibution of different parts to the ducted propeller thrust for 24 considered combinations of the simulation parameters.

Other observations from Fig. 16 and Table 1 are that the increase in the inlet velocity leads to the slight reduction of the thrust coefficient in ducted and unducted propellers. This effect is expected and is due to the increase of the blades drag in the flow direction. With the increase of the rotational speed and the number of blades, the thrust coefficient increases in ducted and unducted propellers. The thrust coefficient grows faster when the number of blades is increased from two to three, than from four to five. In unducted propellers, the difference between the thrust coefficients obtained for two- and five-blade designs is higher than for the ducted propellers.

The figure of merit is a parameter commonly used to measure the efficiency of rotors. Here, the following definition of the figure of merit (FM) is used:

$$FM = \frac{\frac{C_{T}^{3/2}}{\sqrt{2}}}{\frac{\kappa C_{T}^{3/2}}{\sqrt{2}} + \frac{\sigma C_{d0}}{8}}.$$

A comparison between the thrust coefficient for propellers with and without the duct is shown in Fig. 16. The results demonstrate that the thrust coefficient is higher in unducted propellers. One of the reasons is that the current ducted propeller design has not yet been fully optimized. Another reason is that when the duct is included in the propeller design, the length of blades is reduced by a 10%. This reduction of the blade length leads to a lower thrust produced. In the future designs, the blade length reduction should be avoided or mitigated by other design means.



Figure 15. The average contribution of the blades and the duct to the thrust.

(1)

The figure of merit is shown in Fig. 17. The figure demonstrates that unducted propellers are again more efficient.

Figure 18 compares the flow structure obtained with three combinations of simulation parameters that resulted in the close value of the thrust coefficient. The first combination of the simulation parameters is for the two-blade unducted propeller with the rotational speed of 50 rps and the inlet velocity of 1 m/s (Simulation 3 in Table 1). This simulation gave the thrust coefficient of 0.0192. The other two combinations of the simulation parameters gave the thrust coefficient of ~1% larger: the four-bladed ducted propeller with the rotational speed of 50 rps and the inlet velocity of 1 m/s (Simulation 15) and the five-bladed ducted propeller with the rotational speed of 30 rps and the same inlet velocity (Simulation 19).



Figure 16. Comparison of the thrust coefficient obtained for ducted and unducted (isolated) propellers in simulations with different combinations of simulation parameters as described in Table 1.



Figure 17. The figure of merit versus the blade-loading coefficient, $C_T^{3/2}/\sigma$, for ducted and unducted propellers with the different number of blades. Results were obtained for the induced power factor κ =1.15 and the section zero-lift drag coefficient.

On the left side of Fig. 18, velocity fields obtained in Simulations 3, 15, and 19 are shown. Velocity profiles are given on the right side of the figure. As the figure demonstrates, the wake structure is different for ducted and unducted propellers. The velocity magnitude in the wake behind the two-blade unducted propeller is higher than in the wakes of ducted propellers. The area of higher velocity is also larger in the wake of unducted propeller. The velocity induced by the ducted propeller is higher close to the rotational plane (Fig. 18b), but it decreases faster than in the unducted propeller wake. Figure 18c shows significantly weaker wake structure than in Figs. 18a and 18b. The main reason for this is lower rotational speed used in Simulation 19.

To conclude, in order to obtain a similar thrust coefficient, a ducted propeller needs to have more blades to compare with an unducted propeller. If a specific thrust coefficient is required with the low intensity wake, a ducted propeller with high number of blades and lower rotational speed would be preferred.

In Figure 19, streamlines for the two-blade propellers, ducted and unducted, are shown. The velocity downstream the unducted propeller is higher, but the flow structure is more simple than for the ducted propeller.

IV. Conclusion

It has been found that for considered combinations of simulation parameters such as the number of blades, rotational and inlet velocities, the duct can contribute to the thrust up to 30%. Among the considered designs of the ducted propeller, the two-blade ducted propeller had the highest figure of merit of 76 % at the rotational speed of 50 rps and the velocity inlet of 1 m/s.

When comparing with unducted propellers, ducted propellers develop a less intense, but more complicated wake structure. This could lead to the noise reduction and improved aerodynamic performance. However, unducted propellers have shown to have higher thrust coefficient and figure of merit than ducted ones. For the performance comparable with an unducted propeller, more blades should be used in a ducted propeller.

Further optimization of the ducted propeller design is necessary. In particular, the blade length reduction in ducted propellers should be mitigated. The blade twist effect and that of the angle of attack of the blades and the duct should be analyzed.

Acknowledgments

A part of the material is based upon work supported by NASA under award NNX12AJ61A. Computations were conducted using high-performance facilities of the Center for Advanced Research Computing at the University of New Mexico. The authors also would like to acknowledge support from CD-adapco for providing Star-CCM+ for academic purposes.



Figure 18. The wake structure in simulations with the close values of the thrust coefficient: a) $C_T = 0.0192$ in the two-blade unducted propeller with Ω = 50 rps and V_{∞} = 1 m/s; b) $C_T = 0.0194$ in the four- blade ducted propeller with Ω = 50 rps and V_{∞} = 1 m/s; and c) $C_T = 0.0194$ in the five-blade ducted propeller with Ω = 30 rps and V_{∞} = 1 m/s.



Figure 19. Streamlines for the two-blade a) unducted and b) ducted propellers.

References

¹Moller, P. S., "Airborne Personalized Travel Using Powered Lift Aircraft," AIAA 98-5533, *AIAA and SAE World Aviation Conference*, Anaheim, CA, 1998.

²Yilmaz, S., Erdem, D., Kavsaoğlu, M. Ş., "Effects of Duct Shape on a Ducted Propeller Performance", *The 51st AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*, Grapevine, TX, 2013.

³Bulaga, R., "Ducted Fan Efficiency and Noise," SAE Technical Paper 2005-01-3186, 2005, doi:10.4271/2005-01-3186.

⁴Logi AeroSpace. Airborne and Terrain Transport Vehicle. Air Driving in a Tyrannos Air. URL: http://www.logiaerospace.com ⁵FuVeX company. ETSI Navales. Av. Arco de la Victoria, 4. 28040 Madrid- Spain.URL: http://www.fuvex.com

⁶NACA, National Advisory Committee for Aeronautics. URL: www.airfoildb.com/foils/325

⁷Leishman J. G. Principles of Helicopter Aerodynamics. Cambridge Aerospace Series, Cambridge, UK, 2006.

⁸Star-CCM+. Computational fluid dynamics solver, Software Package, Ver.8.06.007, CD-adapco.

⁹CATIA.Computer Aided Three-dimensional Interactive Application, CAD commercial software, V5-6R2013, Dassault Systemes.

¹⁰Host, T. L., and Pulliam, T. H., "Overset Solution Adaptative Grid Approach Applied to Hovering Rotorcraft Flows," 27th AIAA Applied Aerodynamics Conference, San Antonio, TX, 2009.

¹¹Andrew M., Wissink, S. Kamkar, and Pulliam, T. H., "Cartesian Adaptative Mesh Refinement for Rotorcraft Rake resolution," The 28th AIAA Applied Aerodynamics Conference, Chicago, IL, 2010.

¹² Host, T. L., and Pulliam, T. H., "Optimization of Overset Solution Adaptative Grid Approach

Applied to Hovering Rotorcraft Flows," American Helicopter Society Aeromechanics Specialists' Conference, San Francisco, CA, 2010.

¹³Menter, F. R., "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications," AIAA Journal, Vol. 32, No. 8, August 1994, pp. 1598-1605.