The International Journal of Materials and Engineering Technology (TIJMET) (2022) 5(1)
http://dergipark.gov.tr/tijmet
ISSN: 2667-4033

RESEARCH ARTICLE

Effect of propellers numbers and horizontal distance in design of VTOL

Mustafa Varki*[1], Eyüp Yeter[1], M. Hanifi Doğru[2]

[1]Gaziantep University, Aeronautics and Aerospace Faculty, Aeronautics and Aerospace Department, Gaziantep, Turkey
[2]Gaziantep University, Aeronautics and Aerospace Faculty, Pilotage Department, Gaziantep, Turkey

Article Info

Article history:
Received 16.11.2021
Revised: 29.12.2021
Accepted: 02.03.2022
Published Online: 24.03.2022

Keywords:
VTOL
Propeller
Aerodynamic Parameters
Flow Analysis

Abstract

Nowadays, unmanned aerial vehicles with vertical take-off and landing (VTOL) capabilities are increasing. One of the reasons is that these vehicles can take off and land even in difficult conditions and does not need any runway. In design stages of unmanned aerial vehicles, certain design criteria are taken into consideration. VTOL vehicles, which are divided into tilt-wing or tilt rotor, used in areas such as operation areas, efficiency and cost advantage. In this study, CFD analysis is performed to determine the optimum configuration of the VTOL vehicle by considering the horizontal distance between the propellers and the number of propellers of a tilt-wing unmanned aerial vehicle. For this aim, effect of aerodynamic parameters such as thrust, velocity etc. of propellers are investigated. As a result of this study, the suitable propeller position on the wing and number of propellers for the VTOL vehicle are determined.

1. Introduction

The development of Unmanned Aerial Vehicles (UAV) and Vertical Takeoff and Landing (VTOL) for civil and military purposes has shown a significant increase especially in recent years. Many sectors such as agriculture, photography, education seem eager to take part in this developing technology, which increases productivity and minimizes risks due to its unmanned nature. It is critical to design a high-performance UAV or VTOL in order to fulfill the missions effectively and efficiently. The performance of these vehicles are also greatly affected by the efficiency of the propellers. In aviation, a propeller converts the rotational motion obtained from an engine or other power source into thrust. Due to the pressure difference between the forward and rear surfaces of the airfoil-shaped blade, a fluid (such as air or water) is accelerated behind the blade [1]. In addition to being most critical parts for the propulsion system, propellers are simple assemblies except for a few specific details. These details include the hub, which is located in the center and holds the propellers together, and blades whose efficiency varies according to the structure and number of airfoils used. Two-blades propellers are usually used in UAV, considering the stability, weight of the aircraft and the thrust required for take-off. Proper propeller selection for an unmanned aerial vehicle should not be made without considering several factors that characterize propeller performance. There are 3 different methods used to analyze the performance of a suitable propeller and to compare it with the information obtained. The first of these, the analytical method, provides precise solutions that are more time-consuming and sometimes impossible. Also, analytical method doesn't follow any algorithm to solve a problem. On the other hand, the second method, the numerical method is a complete and explicit set of procedures with computable error estimates for solving a problem and it gives approximate solutions with allowable tolerance, less time and more cases. Although there are different numerical methods for solving different types of problems for ordinary differential equations and partial differential equations, the most popular and preferred is the Finite Element Method (FEM). This method is especially used in structural statics, heat transfer, fluid mechanics, mass transfer and electrical potential problems. FEM essentially breaks up a large system into smaller components called finite elements and this is accomplished by discretizing the space in which the equation is solved and dividing it into smaller regions. The experimental method, which is the last method, reaches the real values better, but it may be difficult to reach some experimental setups such as the wind tunnel, and it is affected by instrumental and random errors. As a result, each of these methods has advantages and disadvantages over each other, so they are all used in different phases during studies. In the literature, most of the studies have focused on marine propellers. Studies in the field of aviation remained more limited. Baskaran et al. [2] investigated the computational flow analysis of the marine propeller. Catia was used for a design model and CFD was used for flow analysis. Comparative simulations were performed on the same propeller model using 60 degrees and 45 degrees angular blade. Subhas et al. [3] studied the flow and cavitation analysis of the INSEAN E779a model marine propeller. The research was based on the standard K-Є turbulence model with a liquid application volume to capture the interface between liquid and vapor. Comparison between numerical and experimental analysis suggest that applied CFD method manages to provide reliable output. Veeranagouda Patil et al. [4] evaluated the performance of marine propeller in the non-cavitational condition using the CFD approach. The simulation was carried out different advance coefficient. The results has shown that RNG k-epsilon model works better due to turbulence effects near the wall. Kutty and Rajendran [5] performed numerical analysis using small scale propellers operating with a low reynolds number. Moreover, this analysis was carried out on the slotted propeller

Corresponding Author: Mustafa Varki
e-mail: mvarki@gantep.edu.tr

How to cite this article:
Varki, M., Yeter, E., Doğru, M.H., Effect of propellers numbers and horizontal distance in design of VTOL, The International Journal of Materials and Engineering Technology (TIJMET), 2021, 5(1): 23-27

and these slots were located at seven different positions (12.5%, 25% ,32,5% ,50% ,62,5% ,75% ,87,5%) according to the chord length. The flow domain is divided into 2 different regions as the stationary region and the rotating region that includes the propeller. This domain is created at a distance of 4D from the origin of the propeller to inlet zone, likewise 4D from outlet zone. The standard k-ω turbulence model was applied as it is suitable for the analyzes performed at low Reynolds number. Jeong Hwa Seo et al. [6] managed a study to examine the performance of ship propeller by using ANSYS Fluent. . Flow domain is determined as 10D upstream and 27D downstream. A total of 1.43 million unstructured mesh were created, of which 756,000 hexahedron cells and 538,530 tetrahedron cells. The result shows that there is a maximum discrepancy of 4.67% and 7.12% for the thrust coefficient and the power coefficient, respectively. Göv [7] investigated the effect of distance between the rotors and the number of blades (2,3 and 4 blades) on thrust, torque and power. Flow analysis was done in Solidworks tool. The results of the analysis show that the torque value increases linearly as the number of blades increases. But on the contrary, thrust value does not increase linearly like torque value. Also, Doğru et al. [8] conducted an experimental study to examine the effect of the ducted fan considering the hovering concept of the helicopter. Unlike computational analysis, different techniques such as the static tapping and spring system were used to calculate thrust, and results were close to each other. Hong Fan Wen et al. [9] conducted a numerical analysis using the Ansys fluent tool to determine the performance of the ship propeller blade. Flow domain is arranged to be 5D upstream distance and 10D downstream distance. Total of 4.5 million mesh elements were created. The results show that an accurate result was obtained with a maximum deviation of 6%. Zhang et al. [10] presented aerodynamic analysis of propeller which is used in small aerial vehicles. Flow analysis was done in CFD Fluent tool and sliding mesh method was used. Moreover, both vertical and horizontal wind speeds are indicated and their effects are studied. As a result of the analysis, it is seen that the thrust value of the propeller decreases as the vertical wind speed increases. In addition, it is determined that the aerodynamic performance of the propeller is better at 3500, 4000, 500, 5500 rpm. Oktay et al. [11] investigated the analysis of Quadrotor UAV propeller based on airspeed and propeller thrust coefficient. Numerical and experimental results of the thrust coefficient showed that there is a reasonable discrepancy between %14-%18. Additionally, the thrust coefficient reaches small values at higher airspeeds as expected.

After the literature review, it is seen that most of the studies in the propeller field have focused on the marine propeller. Aero propeller studies remained limited compared to marine propeller. Moreover, the effect of the horizontal distance of propeller and propellers number for VTOL or UAV's performance is not investigated in any studies.

2. Materials and methods

In general, the diameter of the propeller used in unmanned aerial vehicles should not be 24 inches or more. Therefore, advanced precision composites (APC) propeller 15x4W is used due to its datasheet availability and the fact that it is a common used propeller in UAV's. In this study, performance analysis is made by changing the number of propellers and the horizontal distance between these propellers. As stated in the previous section, there are 3 different methods for performance analysis and CFD method is applied in this analysis. The experimental method will also be added in future studies.

2.1. Propeller Model

The APC 15x4W propeller used in this study has 2 blades, fixed pitch angle, and 0.379 diameter. The operating range is between 1000-10000 rpm. The propeller is formed of thin-thickness airfoil profile suitable for a low reynolds number as demonstrated in Figure 1.

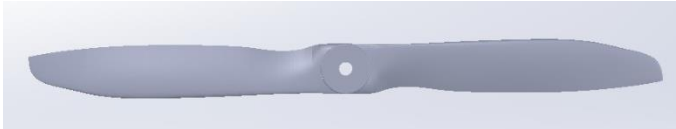


Figure 1. APC 15 inches x 4 inches propeller blade

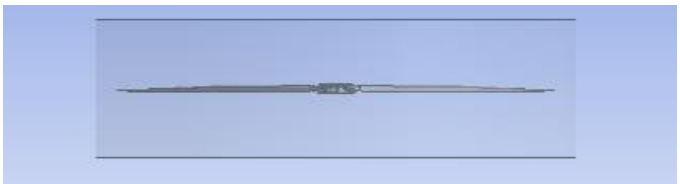
Also, the main parameters of the APC 15X4 propeller are indicated in Table 1.

Table 1. Dimensions of APC 15x4 propeller.

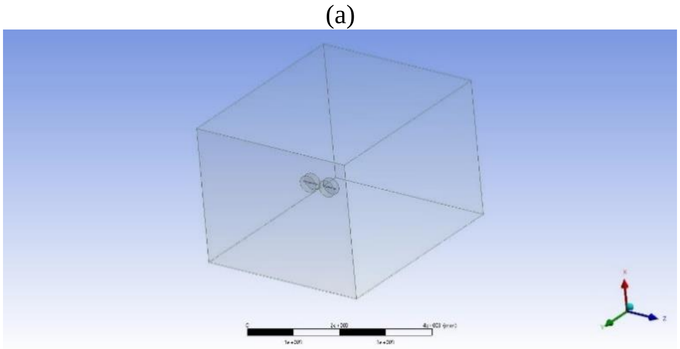
Parameters	Dimensions
Diameter	381 mm
Pitch	101.6 mm
Weight	70.0233221 g
Hub Diameter	28.702 mm
Hub Thickness	10.668 mm

2.2. Computational Parameters

This study focuses on the performance analysis of the propellers using ANSYS FLUENT and their effects on each other depending on the horizontal distance and propellers number. The Multiple Reference Frame (MRF) model approach is selected to give the propeller rotation effect in flow analysis. The flow domain is divided into two domains: a stationary domain and a rotating domain. Rotating domain is created as a cylinder, completely enclosing the propeller and hub region. The stationary domain is also designed as a square to include all regions. These two domains along with their dimensions are shown in Figure 2 with a single propeller.



(a)



(b)

Figure 2. Flow domain (a) rotating domain (b) stationary domain

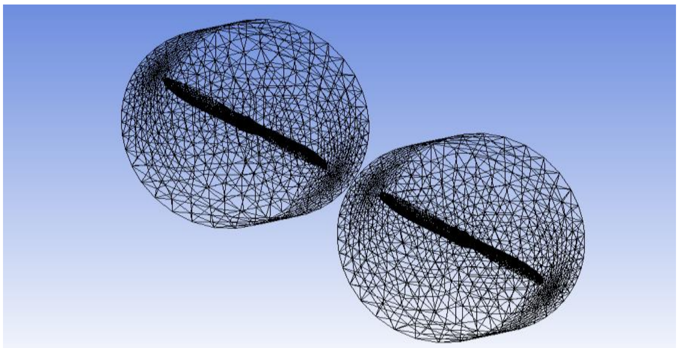
In analyzes made using the MRF method, inlet and outlet distances should be adjusted appropriately in order to distribute the flow more smoothly and to obtain more consistent results. At close boundary distances, the convergence problem occurs due to the recirculation of the flow. In this study, given that the diameter of the propeller is D, the diameter of the rotating domain is set to 1.1D and the thickness to be 0.4D. The stationary domain is also created in a rectangle shape with inlet distance is set to be 4D and outlet distance is set to be 8D according to origin of the propeller.

2.3. Mesh Generations

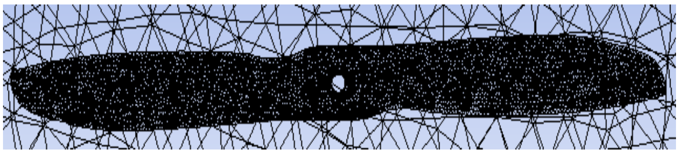
Meshing plays an important role when it comes to analysis processes. Creating the optimal mesh is the foundation of analysis and flow simulations because the mesh affects the accuracy, convergence, and speed of the analysis [12]. In this study, cell sizes in the mesh along the blades of the propeller and in the rotating domain are smaller than in the stationary domain. Fully tetrahedral unstructured mesh is implemented for both the rotating and stationary flow domain. Unstructured mesh was preferred because it is easier and faster in complex geometries. Total numbers of cells generated are 350 elements for each case. Figure 3 shows the mesh on the propeller surface and table 2 shows the detail for mesh parameters.

Table 2. Mesh parameters.

Size Function	Curvature
Max Face Size	0.22 m
Defeature Size	0.0011 m
Growth Rate	1.20
Min Size	0.002 m
Max Tet Size	0.22 m
Curvature Normal Angle	18°



(a)



(b)

Figure 3. Surface mesh for (a) rotating domain (b) propeller

2.4. Boundary Conditions

CFD simulations were set up with a fixed free-stream velocity and a fixed rotational velocity for each propeller. On the inlet and outlet boundary, velocity inlet and outflow conditions were applied with turbulence intensity of 0.1%. Turbulence intensity is adjusted according to the wind tunnel intensity measured by [13,14]. No slip conditions were set on the walls because fluid have zero velocity relative to the boundary. As the Multiple Reference Frame(MRF) approach was chosen, rotational frame motion was applied to the rotating domain and a constant speed of 10000 rpm was applied. This approach is best suited for analyzes that will show the interaction between two different regions, stationary and rotating. The wall forming the propeller blade and hub is also designated as rotating at zero speed relative to the adjacent cell region. No rotational velocity or direction is given to the stationary parts. The standard k-ω turbulence model is used throughout the analysis, as it gave better results with lower Reynolds numbers. In addition, k-w turbulence model can solve turbulence parameters very close to the boundary or wall region and it provides more accurate solutions in boundary areas close to the wall. All solver parameters used with MRF method are given in table 3.

Table 3. Solver parameters.

Parameters	Settings
Solver	Transient
Equation of State	Constant Pressure
Turbulence Model	Standard k-ω
Propeller Motion Type	Frame Motion
Inlet Boundary	Velocity Inlet
Outlet Boundary	Outflow
Residual Error	$1x10^{-4}$
Pressure-velocity Coupling	Coupled

3. Results and discussion

In this study, the thrust and torque values are obtained at six different cases by using ANSYS Fluent program. In one of these cases, the propeller is analyzed alone, in the other cases, two propellers are used and the distance between them was set as D/3, D/2, D,2D ,3D. While analyzing these cases, the rotational speed and inlet speed are kept constant values (rpm=10000, v=1.02 m/s) and the mesh qualities are kept close to each other. Thrust coefficient and torque coefficient values are also calculated together with the obtained values. These coefficients are explained in Equations (1) and (2). In these equations, ρ is fluid density in standard condition, T is thrust value, n is propeller speed and D is propeller diameter.

$$C_T = \frac{T}{\rho n^2 D^4} \quad (1)$$

$$C_Q = \frac{T}{\rho n^2 D^5} \quad (2)$$

Fig. 4 shows the pressure distribution on a one propeller. As expected, the pressure at the blade tips has very small values. Also, Fig. 5 shows the velocity distribution for D/3, D and 3D. As the distance between propellers increases, the effect of the air coming out of the propeller decreases.

When 1 propeller is analyzed under similar conditions, the thrust value obtained was found to be 34.808 N. When 2 propellers are used and the horizontal distance between them is changed, the values are shown in fig 6. Thrust difference between 3D and D/3 horizontal distances was determined as %2. It has been determined that the variation of thrust decreases after the D distance because the effect of the air flow on the propellers after this distance gradually loses its importance.

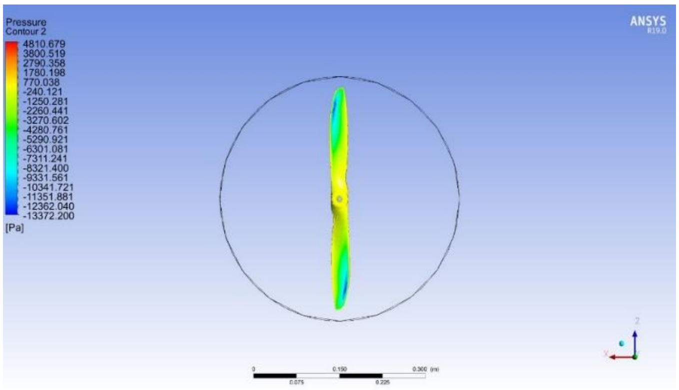
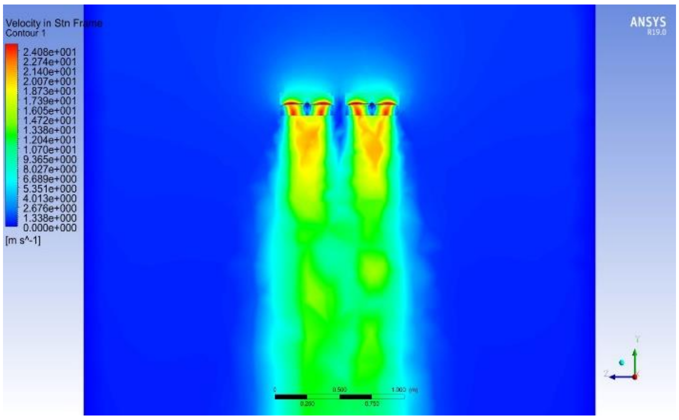
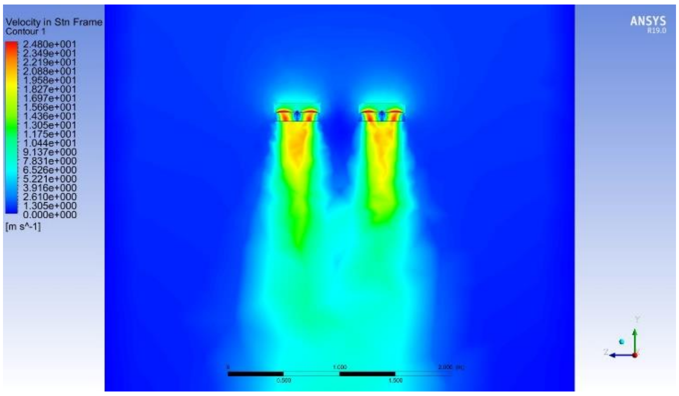


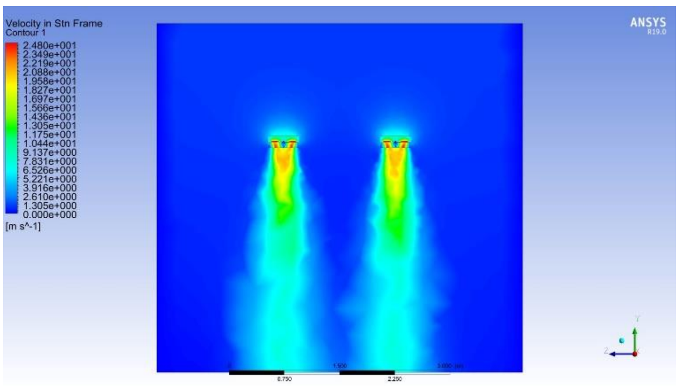
Figure 4. Pressure distribution of propeller.



(a)



(b)



(c)

Figure 5. Velocity distribution of propellers (a)D/3 (b)D/2 (c)D

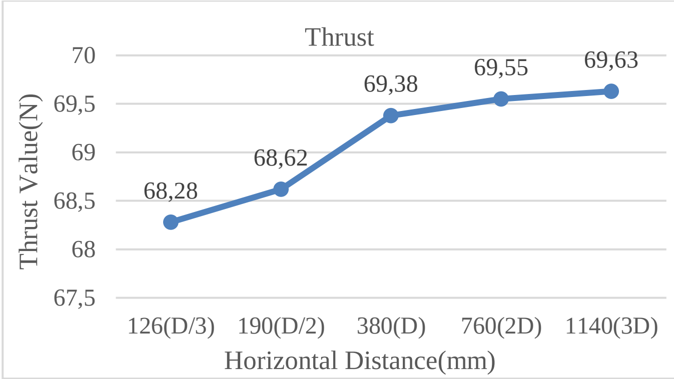


Figure 6. Thrust values according to various horizontal distance

Also, the thrust and torque coefficient values at these distances are given in Table 4 and Table 5. The same similarity according to the increase rate of Thrust is clearly seen in these two values.

Table 4. Thrust coefficient(C_T).

Distance	Values
D/3	0.09624
D/2	0.09672
D	0.09779
2D	0.09803
3D	0.09814

Table 5. Torque coefficient(C_Q).

Distance	Values
D/3	5.459×10^{-3}
D/2	5.482×10^{-3}
D	5.638×10^{-3}
2D	5.704×10^{-3}
3D	5.749×10^{-3}

4. Conclusions

In this study, the effects of the number of propellers and the horizontal distance between the propellers are observed. For the purpose of the study, single propeller and two propellers with their horizontal distances changed is analyzed and thrust, thrust coefficient and torque coefficient values are obtained. Five different cases are examined for horizontal distance. (D/3, D/2, D, 2D, 3D). Velocity and pressure distributions are created according to the given boundary conditions and the established flow domain. As can be seen from the results, while the rate of increase in thrust value up to the D distance is higher, the effect of the propellers on each other decreases after the D distance, and therefore the increase in thrust is relatively less. The thrust obtained by 2 propellers is between D and 2D, very close to the thrust obtained by single propellers. From here, it can be say that these distances should be taken into considerations when more propellers will be used. However, the length of the wingspan determined for the aerial vehicles to be designed is also important and is one of the limitations of the propeller distances.

Author contributions

Mustafa Varki: Analysis, Writing - original draft
Eyup Yeter: Supervision, review & editing
M.Hanifi Dogru: Supervision, review & editing

References

1. The Wikipedia website. [Online]. Available: https://en.wikipedia.org/wiki/Propeller (accessed on 2 November **2021**)
2. Baskaran, S., Dhineshraj, L., Dinesh, J., Dineshkumar, B., Dineshkumar, T., Design and Flow Analysis of Marine Propeller using Computational Fluid Dynamics, International Journal of Research and Scientific Innovation, **2019**, 56-58
3. Subhas, S., Saji, V. F., Ramakrishna, S., & Das, H. N., CFD analysis of a propeller flow and cavitation, International Journal of Computer Applications, **2012,** 55(16)
4. Patil, V., Purushothama, H., Manjunatha, A., & Mishra, V. K., Performance Evaluation of Marine Propeller using Numerical Simulation, Indian Journal of Science and Technology, **2016,** 9(45)
5. Kutty, H. A., & Rajendran, P., 3D CFD simulation and experimental validation of small APC slow flyer propeller blade. Aerospace, **2017,** 4(1), 10
6. Seo, J. H., Seol, D. M., Lee, J. H., & Rhee, S. H., Flexible CFD meshing strategy for prediction of ship resistance and propulsion performance. International Journal of Naval Architecture and Ocean Engineering, **2010,** 2(3), 139-145
7. Göv, İ., Rotor Spacing and Blade Number Effect on the Thrust, Torque and Power of a Coaxial Rotor, El-Cezeri Journal of Science and Engineering, **2020**, 7(2), 487-502
8. Doğru, M. H., Güzelbey, İ. H., & Göv, İ., Ducted Fan Effect on the Elevation of a Concept Helicopter When the Ducted Faintail Is Located in a Ground Effect Region, Journal of Aerospace Engineering, **2016,** 29(1); 04015030
9. Hong, F. W., & Dong, S. T., Numerical analysis for circulation distribution of propeller blade, Journal of Hydrodynamics, Ser. B, **2010,** 22(4); 488-493
10. Zhang, C., Xie, S., & Qin, T., Aerodynamic analysis of small propeller in wind field using CFD, In Proceedings of the 2015 4th International Conference on Sustainable Energy and Environmental Engineering, **2016**
11. Oktay, T., & Eraslan, Y., Computational fluid dynamics (Cfd) investigation of a quadrotor UAV propeller, In International Conference on Energy, Environment and Storage of Energy (ICEESEN 2020) **2020**; 1-5
12. Simutech group website. [Online]. Available: https://simutechgroup.com/why-is-meshing-important-for-fea-fluid-simulations/ (accessed on 2 November **2021**
13. Brandt, J., & Selig, M., Propeller performance data at low reynolds numbers, In 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, **2011**; 1255
14. Deters, R. W., Ananda Krishnan, G. K., & Selig, M. S., Reynolds number effects on the performance of small-scale propellers. In 32nd AIAA applied aerodynamics conference, **2014**; 2151