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

RESEARCH ARTICLE

Investigation of the hub diameter effect on propeller thrust

Enes Coşkun*[1], Mehmet Hanifi Doğru[2]

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

Article Info

Article history:
Received: 16.11.2021
Revised: 10.05.2022
Accepted: 13.05.2022
Published Online: 03.06.2022

Keywords:
Propeller
Computational Fluid Dynamics
Hub Effect
Thrust Distribution

Abstract

With the widespread use of unmanned aerial vehicles in the aviation industry, the importance of detailed examination of propellers, whose task is to provide thrust, has also increased. A propeller is a part that is formed by attaching more than one aerodynamically shaped blade to a hub and produces thrust by being rotated by a motor. The amount of thrust that is produced by a propeller depends on some parameters such as diameter, number of blades, pitch angle, etc. The aim of this study is to investigate the thrust distribution along a propeller diameter section with the gradual increase of the hub diameter. Related studies show that the maximum thrust of a propeller is obtained in the region between 75% and 85% of the propeller length. In order to obtain the necessary data, numerical flow analyzes were made and the results were discussed. As a conclusion, at the very closer to the root of the propeller blade, the amount of produced thrust is considerably low compared to the near tip of the propeller. Therefore, the thrust loss due to the increase of the propeller hub diameter is negligible and maximum thrust is obtained in the expected region.

1. Introduction

In the last few years, the use and popularity of aerial vehicles (especially unmanned aerial vehicles) have increased rapidly. Aircraft operate in a wide range of areas, primarily for transportation and cargo transportation, as well as for civil and military purposes in firefighting, search and rescue, mapping, and sports activities. Among the air vehicles, unmanned aerial vehicles have an important place thanks to the positive benefits they provide. Unmanned aerial vehicles are divided into three categories: fixed-wing, rotary-wing, and hybrid (VTOL). Every air vehicle needs propulsion systems to move vertically or horizontally. In today's unmanned aerial vehicles, generally propeller-engine configuration is used. The propeller is basically a rotary blade with twisted airfoil sections. A simple propeller consists of two parts, the blades, and the hub. The blades convert the rotational movement coming from the engine into the thrust force that will push the aircraft forward/up. The airfoil sections of the blades act like wings and create a pressure difference between the lower and upper portions of the propeller blade as the propeller rotates, thanks to the air they are exposed to. Due to the diameter of the rotating blades, the linear velocity at the tip is higher than the linear velocity at the root. In this case, while the maximum thrust is expected to be at the tip, the maximum thrust occurs at approximately 80% of the blade length due to the losses at the tip [1]. The hub is the part that connects the blades to each other and provides the connection between the propeller and the motor shaft. The main purpose of this study is to investigate how much the thrust force is affected by the increase in hub diameter.

The parameters that determine the performance of a propeller such as thrust, and torque are functions of diameter, pitch, rotational speed, and airfoil. There are many propeller designs for different needs of different aircraft. With the widespread use of unmanned aerial vehicles, research on propeller performance and design has gained importance. There are three methods for analyzing the performance of a propeller, these are experimental, analytical, and numerical methods. Analytical methods are advantageous in that they give faster results than experimental and numerical methods, but they are disadvantageous in approaching real values. They are often used to get a preliminary idea during the design phases, due to the time saving provided by their fast results. Experimental tests are the method that gives the most accurate results among these methods. However, there are also disadvantages associated with experimental methods. Although the test equipment such as the wind tunnel used to perform the necessary tests are not easily accessible, uncertainties arise from the equipment or the test setup during the test procedure. Numerical methods are the method in which differential flow equations are solved in order to calculate values such as pressure, temperature, and velocity by modeling the flow field in the computer environment. Although there are different methods for solving these differential equations, the most popular method is the Finite Element Method. By dividing the flow field into very small elements with a mesh structure, these equations are solved for each element and the results are combined to reach the overall result. In order to perform computational fluid analysis, it is necessary to model the flow field very well and to provide the necessary processing power for large amounts of calculations. All three methods for propeller performance analysis can be used for different needs in different parts of the design phase.

Haidar et al. [2] performed a performance analysis of a propeller by using analytic methods with varying alpha design parameters and applied two-stage scoring to select the best design. The design with the highest score among 42 alpha designs was selected as the best. As another analytical approach, Benini [3] used Combined Momentum Blade Element Theory (CMBET) for light and moderately loaded marine propeller and

Corresponding Author: Enes Coşkun
E-mail: nscoskun@gantep.edu.tr

How to cite this article:
Coşkun, E., Doğru, M.H., Investigation of the hub diameter effect on propeller thrust, The International Journal of Materials and Engineering Technology (TIJMET), 2022, 5(1): 43-47

compared with the CFD solution. The result of the comparison shows that the accuracy of such predictions is very sensitive to advance ratio and CMBET method is accurate only when the three-dimensional effects are of secondary importance. On the other hand, three-dimensional Reynolds-Average Navier-Stokes (RANS) method gives more accurate results independent of the advance ratio with a maximum discrepancy of 5% when compared with the experimental data.

Kutty et al. [4] made an analysis of advanced precision composites (APC) Slow Flyer propeller blade, using the commercially available CFD solver FLUENT released by ANSYS Inc. to determine the error between the calculated values, and experimental data of the thrust coefficient, power coefficient, and efficiencies. The analysis performed by using the k- ω turbulence model and Multiple Reference Frame method is used to simulate the rotation effect of the propeller for analysis. The flow domain consists of two parts, a stationary domain, and a rotating domain. Five mesh resolutions (standard, coarse, mid, mid-fine, fine) were applied for the error estimation of performance values from the experimental data. The results showed under-prediction for low advance ratios for both thrust coefficient and power coefficient and showed over-prediction for high advance ratios for both efficiency and power coefficient. Varki et al. [5] also used Multiple Reference Frame method to analyze the effect of the propeller number and the horizontal distance in Vertical Take-Off and Landing.

Investigation of the performance of the propellers is not only the research area of aerospace. Propellers are also used in the marine field to give thrust to marine vehicles. Kumar et al. [6] investigated the carbon composite marine propeller in the aspect of strength, aerodynamic loading, and static loading conditions. They designed the propeller and performed CFD analysis by using RANS equations solver StarCCM+. Thrust distribution along one blade span result shows that the maximum thrust occurred around 85% of the propeller radius. They performed a static loading test by dividing the blade into six sections ranging from 0.4R – 0.9R for deflection measurement.

Ducted fans are basically propellers that are mounted within a cylindrical shroud. They are used because of improving the propulsive performance comparing the open propellers. Akturk et al. [7] performed CFD analysis by using general purpose code Ansys-CFX and they compared the results with the available wind tunnel test data for disc loading and exit pressure for validation. One of the findings emphasized in the paper is the effect of the hub shape. Separation from the hub corner is an important source of loss. Also, Doğru et al. [8] prepared an experimental setup for a ducted fan that inside the ground effect region, obtained thrust forces with two different methods, static tapping, and spring method. Besides the ducted fans, coaxial rotors are being a popular research area in the propulsion field. Coaxial rotors are basically two rotors that rotate in opposite directions and are located as up and down on the same axis. Göv [9] investigated the effect of coaxial rotors on each other and calculated the power, thrust, and torque values with 6 different rotor spacing in SolidWorks flow simulation. Tang et al. [10] established an optimization approach for Contra-Rotating Propellers (CRP) by considering the influence of blade numbers, diameter, rotational speeds, axial distance, and torque ratio.

Experimental analysis of the performance of propellers has significant importance because of its reliable results. Brandt et al. [11] performed subsonic wind tunnel tests with 79 propellers that are produced for UAV applications in 9 to 11in diameter range. They tested the propellers with varying wind tunnel speeds until reaching the windmill state (zero thrust). The results will provide a large database that can be used for selecting the appropriate propellers for a wide variety of applications. The reliability of a CFD analysis is strictly dependent on the quality of the mesh structure and the correct definition of the domain. Generally, the domain dimensions are selected far away from the propeller in a way it cannot affect the flow by estimating. Islam et al. [12] studied a mesh and domain optimization strategy for CFD technique. They used commercial RANS solver StarCCM+ to predict the propulsive performance at different conditions. Six input domain and meshing parameters such as outer diameter, inlet distance from propeller center, outlet distance from propeller center, refinement zone diameter, refinement zone extent, and mesh base size with their ranges were used to perform optimization. Boundary conditions are assigned as velocity inlet and pressure outlet for each simulation. The optimized mesh and boundary performed six times faster than the popular mesh and domain configuration.

2. Materials and methods

As mentioned in the first section, there are three methods used for propeller performance analysis. In this study, CFD method, which is a numerical method, was used. Three-dimensional analysis and numerical simulations were performed with the commercially available CFD solver ANSYS Fluent. The necessary information about the method and analysis is further explained in the following sections.

2.1. Geometry

The propeller used in this study is the APC15x4w propeller (Figure 1) manufactured by Advanced Precision Composites with a diameter of 15 in (381mm) and a pitch of 4 in (101.6mm). This propeller has features suitable for use on unmanned aerial vehicle platforms. Each section of this propeller, which has a variable pitch angle across the blade, is modeled using the NACA4412 airfoil. To explain the pitch angle simply, it behaves the same as the amount of penetration of the screw into the target area with each turn of a screw. Therefore, the airfoil angle varies along the diameter of the propeller. Generally, thick, and high-angle wing profiles are used in the near hub part, while the wing profile angle decreases towards the tip and thin profiles are used near tip region. One of the reasons for using thick profiles in the near hub part is to increase the strength.

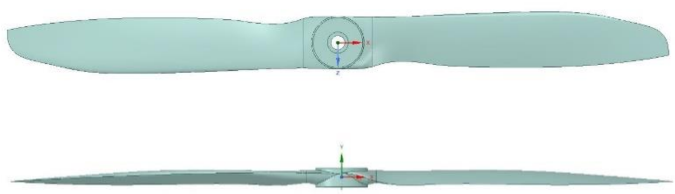


Figure 1. APC 15x4w propeller

2.2. Numerical Analysis Setup

In order to perform a flow analysis in a CFD simulation program, both the solid parts and fluid domains must be determined correctly. In this study, the Multiple Reference Frames (MRF) method was used to simulate the air passing over the propeller. Therefore, two different flow regions, rotating and stationary, were created (Figure 2). In order to simulate the rotational effect of the propeller, the rate of revolution is defined in the rotating flow domain and the propeller is defined as stationary. The literature review shows that in CFD analyzes of propeller performance when modeling the outer flow region, the distance to the front region (upstream) is between 3 and 6 times the diameter of the propeller, and the distance to the rear region (downstream) is between 4 and 10 times the propeller diameter [12]. In this study, the outer domain is modeled as 4D upstream and 8D downstream as shown in Figure 2. For the rotating region, 1.1D diameter and 0.4D width are defined. By modeling the flow region in this way, proper upstream and downstream flow formations of the propeller are provided. Thus, convergence problems caused by incomplete flow regimes are prevented.

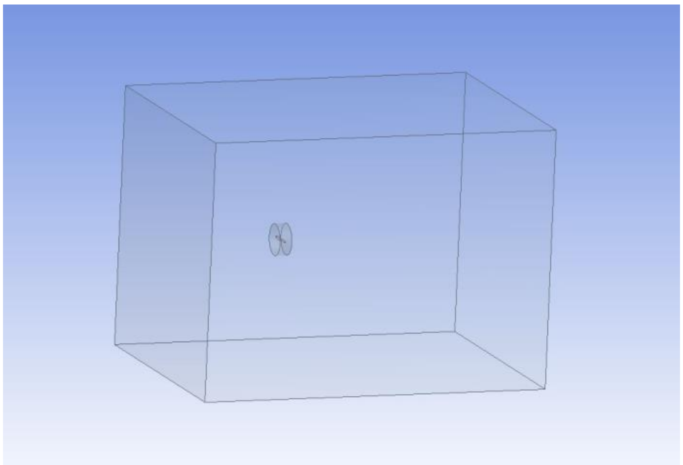
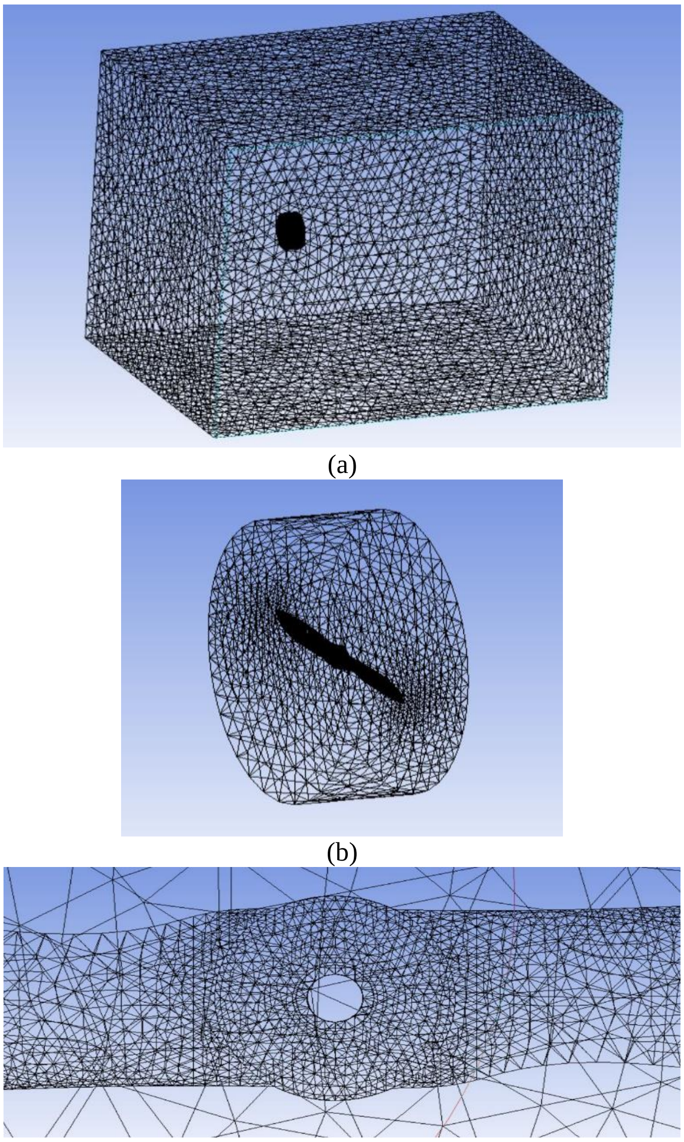


Figure 2. Outer flow domain

Meshing, which is the next step after defining the flow domain, is a very important step in CFD analysis. In fact, the accuracy of CFD analysis is highly dependent on the quality of the mesh structure. In this study, tetrahedral mesh elements, which are very successful in modeling curved and complex geometries such as propellers, were used. By making the necessary definitions and refinements, an unstructured tetrahedral mesh grid was created consisting of smaller elements near the propeller and coarsening as it moves away from the propeller (Figure 3). In this study, 4 different analyzes were made depending on the change of hub diameter, and the mesh structures that were created for each geometry have approximately the same orthogonal quality, skewness, and mesh element number values.

The boundary conditions of the analysis represent the real physics of the flow model in the analysis domain. All the definitions such as inlet, outlet, rotations, axes, etc. are defined in this section. The flow characteristic is modeled as pressure-based, transient. In this study, velocity inlet and outflow condition are defined for transient flow analysis as shown in Table 1. In order to model the rotation effect of the propeller, multiple reference frame (MRF) approach is used by defining the 10000 rpm (revolutions/minute) which is equal to 166,667 rps (revolutions/second) to the rotating zone frame in the axis of rotation. Interface is defined to the connection faces of the rotating and stationary boundaries for mesh interaction. The aim of the interface is to transfer the calculated values inside one domain to another domain. For the turbulence model, standard k-ω model is used in order to predict the turbulence effects accurately. Kutty et al. [4] performed 3 different turbulence models (standard k-ε, standard k-ω, and SST k-ω) in the study and found that the k-ω model gives more accurate results when compared with the other models. Methods for solving the flow equations are assigned as follows, coupled method for pressure-velocity coupling scheme, least squares cell based for gradient and second-order upwind scheme for both momentum, turbulent kinetic energy, and specific dissipation rate. These settings are applied to each case study in the same way.



(a)

(b)

(c)

Figure 3. Unstructured tetrahedral mesh structures (a) Outer domain (b) Rotating domain (c) Propeller

Table 1. Boundary conditions and solver parameters.

	Boundary conditions and parameters
Solver	3D, Transient
Turbulence Model	Standard $k - \omega$
Fluid	Air
Inlet	Velocity inlet
Outlet	Outflow
Propeller	No slip wall
Rotating Domain	Rotating frame motion

3. Results and discussion

As mentioned in the first section, the parameters that define the performance of a propeller are thrust and torque. These parameters are the definition of force and moment around the propeller. The calculated thrust and torque coefficients for each case are shown in Table 2. Thrust and torque coefficients are calculated using the following equations with $T\ [N]$ is thrust, $Q\ [Nm]$ is torque, $\rho\ [kg.m^3]$ is air density, $n\ [rps]$ is rotational speed, and $D\ [m]$ is diameter.

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

$$K_Q = \frac{Q}{\rho n^2 D^5} \quad (2)$$

Table 2. Coefficients of thrust and torque.

Hub diameter with respect to propeller diameter	K_T	K_Q
0.08D	0.0493	0.00276
0.09D	0.0491	0.00275
0.10D	0.0490	0.00274

One of the main aims of this study was to investigate the thrust distribution along the blade. It is important to determine the loadings on the propeller blade for use in modal and structural analysis. Thrust distribution graph provides the necessary information for these studies. In Figure 4, the thrust distribution graph is shown with respect to the blade section that is defined with r/R. The section radius is represented by r while the propeller radius is R. The propeller radius is divided into 30 sections and the thrust values at each section are plotted in the graph.

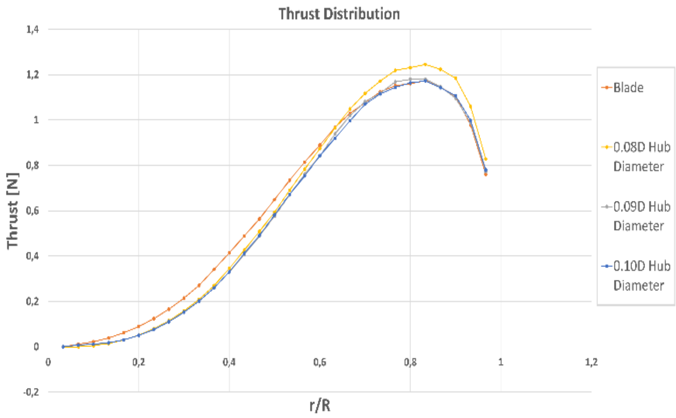
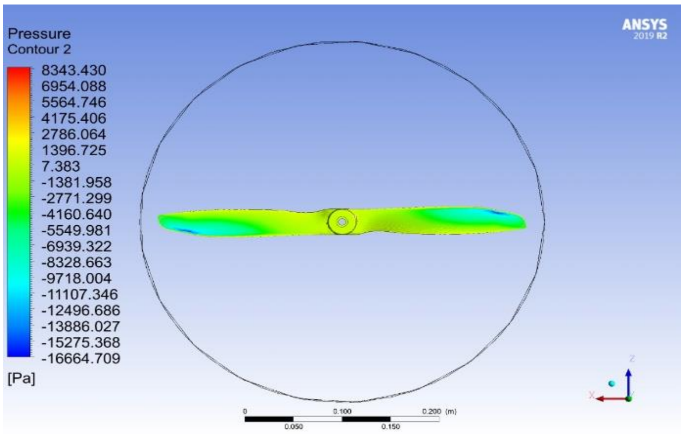
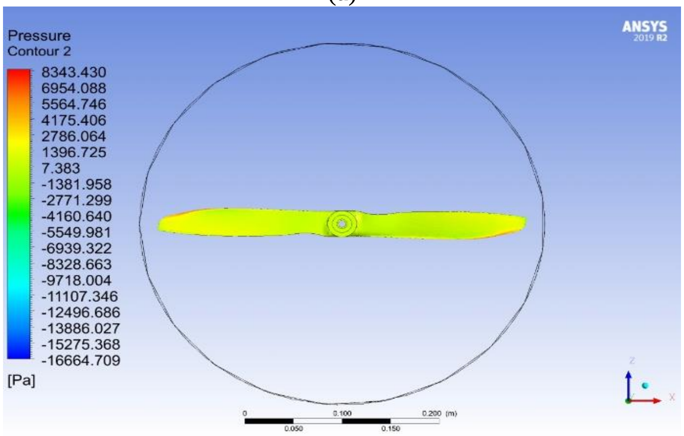


Figure 4. Thrust distribution along the blade section for different hub diameters

As can be seen in Figure 4 the maximum thrust occurs around 0.82 of the propeller radius. The result and thrust distribution graph obtained in Figure 4 agree with the results obtained in Ismail et al. [1] and Kumar et al. [6]. As the section moves away from the 0.82 point, a decrease in thrust value is observed with the effect of the losses caused by the tip vortices. Figure 5 shows that minimum pressure also occurs at approximately 0.82 of the propeller. This negative pressure region creates maximum thrust. It is calculated that nearly 0.6% of the total thrust is generated at the 0-0.2r/R region, while 66.1% of the total thrust is generated at the 0.2-0.8r/R region, and 33.3% is generated at the 0.8-1r/R region. Based on these results shown in Table 3, the thrust produced in the region of 0-0.2 is negligible.



(a)



(b)

Figure 5. Pressure distribution contour of the propeller (a) Upper surface (b) Lower surface

Table 3. Thrust rates by regions.

	Hub Region (0 – 0.2 r/R)	Intermediate Region (0.2 – 0.8 r/R)	Tip Region (0.8 – 1 r/R)
Ratio of generated thrust to the total thrust (%)	0.6	66.1	33.3

4. Conclusions

The 3D CFD simulation of a 15 in propeller was performed by using the commercially available CFD solver ANSYS Fluent in order to investigate the performance characteristics of the propeller, hub diameter effect to thrust, and thrust distribution along the blade section. Necessary domains and conditions for simulations are used appropriately. MRF approach is used to simulate the rotational motion of the propeller and standard k-ω turbulence model is used according to the literature review. The thrust distribution results showed that very little amount of thrust is produced near hub region, and it is relatively negligible when compared with the intermediate and tip regions. This low-thrust region can be either optimized to increase the thrust or can be used for improving strength or other purposes. The maximum thrust occurs near 82% of the propeller blade as expected. This region can be shift if needed by changing the propeller geometry variables such as blade angle in the design process. The results also showed that the hub diameter effect on the thrust is also negligible while it is in the low-thrust region.

Author contributions

Enes Coşkun: Analysis, Writing - original draft
M.Hanifi Dogru: Supervision, review & editing

References

1. Ismail, K.A., Rosolen, C.V., Effects of the airfoil section, the chord and pitch distributions on the aerodynamic performance of the propeller. Journal of the Brazilian Society of Mechanical Sciences and Engineering, **2019**, 41(3): 1-19
2. Haidar, M.D.F., Moelyadi, M.A., Hartono, F., Design and Performance Analysis of Low Reynolds Number Propeller using Analytical Methods by Varying Blades Alpha Design. In IOP Conference Series: Materials Science and Engineering, IOP Publishing, **2019**, 645(1): 012021
3. Benini, E., Significance of blade element theory in performance prediction of marine propellers. Ocean engineering, **2004**, 31(8-9): 957-974
4. Kutty, H. A., Rajendran, P., 3D CFD simulation and experimental validation of small APC slow flyer propeller blade. Aerospace, **2017**, 4(1): 10
5. 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, **2022**, 5(1): 23-27
6. Kumar, A., Krishna, G.L., Subramanian, V.A., Design and analysis of a carbon composite propeller for podded propulsion. In Proceedings of the Fourth International Conference in Ocean Engineering (ICOE2018), Springer, Singapore, **2019**, 203-215
7. Akturk, A., Camci, C., A computational and experimental analysis of a ducted fan used in VTOL UAV systems. Department of Aerospace Engineering, Pennsylvania University, USA, **2011**
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. İbrahim, G., 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
10. Tang, J., Wang, X., Duan, D., Xie, W., Optimisation and analysis of efficiency for contra-rotating propellers for high-altitude airships. The Aeronautical Journal, **2019**, 123(1263): 706-726
11. 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
12. Islam, M. F., & Jahra, F., Improving Accuracy and Efficiency of CFD Predictions of Propeller Open Water Performance. Journal of Naval Architecture & Marine Engineering, **2019**, 16(1)