CHEMICAL ENGINEERING TRANSACTIONS
VOL. 76, 2019

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Petar S. Varbanov, Timothy G. Walmsley, Jiří J. Klemeš, Panos Seferlis

**ISBN** 978-88-95608-73-0; **ISSN** 2283-9216

DOI: 10.3303/CET1976031

# Aerodynamic Analysis of a Quadcopter Drone Propeller with the Use of Computational Fluid Dynamics

Konstantinos Christodoulou, Michail Vozinidis, Asterios Karanatsios, Efstathios Karipidis, Fotios Katsanevakis, Zinon Vlahostergios*

Democritus University of Thrace, Xanthi, Greece
zvlachos@pme.duth.gr

In the current work, an analysis for the initial aerodynamic assessment of a small sized quadcopter drone propeller is presented, focusing on the drone aerodynamic characteristics optimization. The objective of the work is to design and construct from scratch a quadcopter drone, in order to participate to the UAS Challenge 2020 student competition (www.imeche.org/events/challenges/uas-challenge). The study of the quadcopter design and construction is carried out by the Democritus Aeronautical Rescue Team (DART), a team that consists of students of the Democritus University of Thrace School of Engineering. The performance analysis which is presented is mainly based on the aerodynamic characterization of an initial selected propeller design, which will be used for the examined quadcopter, with the use of Computational Fluid Dynamics (CFD). Three turbulence models were selected for the computations each one based on a different modelling approach. The computations are assessed and conclusions regarding the propeller design are derived. Based on the computations, the optimal propeller shape will be designed, and the results will further help to the refinement of the overall quadcopter aerodynamic layout in the final detailed design stages for the UAS Challenge competition.

## 1. Introduction

During the last decade there has been a huge development in the designing and evolving technologies based on their integration with unmanned aerial systems/drones. These integrated drone systems cover a large range of applications that can greatly help humanity in every aspect. For instance, they can be used for surveillance purposes for predicting physical disasters, they can provide services for precision agriculture (Sarghini and De Vivo, 2018) and cartography, aid in humanitarian missions and greatly assist rescue missions. In general, these systems can be used for monitoring and accessing by air places difficult to be reached by other means.
The current work presents the initial steps in the designing procedure of a quadcopter drone in order to participate to the IMECHE UAS Challenge 2020 competition (www.imeche.org/events/challenges/uas-challenge), by the Democritus Aeronautical Rescue Team (DART). The DART team consists of students of various Engineering Faculties of Democritus University of Thrace (DUTH). During the project, the DART team must design, analyse and construct an autonomous rescue quadcopter drone in order to complete a desired specific mission. The mission of the drone during the competition is to successfully send and drop first aid packages to targets/coordinates arranged in an automated course which must be followed by the drone, covering a flight distance of 9 to 10 km. After completing its mission, the drone must return to the starting point without any human interference. The overall weight of the drone (including the payload) should not exceed 6.9 kg and it must be capable of tracking and recording specific waypoints during its flight with the use of a GPS device.
The presented study is focused on the aerodynamic performance characteristics assessment of an initially selected propeller for the quadcopter with the use of Computational Fluid Dynamics (CFD) and on the performance assessment of three turbulence models that follow different turbulence modelling methodologies. In the literature there are various papers that use CFD for validating propeller aerodynamic characteristics for unmanned aerial vehicles, manned airplanes and wind turbines. For instance, Yeong and Dol (2016) used the k-ω SST model to optimize the aerodynamic performance of a micro aerial vehicle. Kutty and Rajendran (2017)

Paper Received: 01/04/2019; Revised: 29/05/2019; Accepted: 29/05/2019
Please cite this article as: Christodoulou K., Vozinidis M., Karanatsios A., Karipidis E., Katsanevakis F., Vlahostergios Z., 2019, Aerodynamic Analysis of a Quadcopter Drone Propeller with the Use of Computational Fluid Dynamics, Chemical Engineering Transactions, 76, 181-186 DOI:10.3303/CET1976031

adopted the standard k-ω turbulence model in order to determine a small-scale propeller performance. Salpingidou et al. (2018) implemented the SST k-ω turbulence model in order to derive surrogate models for modelling the flow around a propeller of a small aircraft. All the literature studies that examine the flow around drone propellers, to the authors knowledge, adopt simple linear eddy-viscosity turbulence models combining them with the rotating reference frame approach, or with modelling the actual rotating propeller wall. However, the importance of using more advanced and sophisticated turbulence models for modelling complicated turbulent flows around airfoils and consequently blades, wings and propellers that involve boundary layer transition and flow separation, is highlighted by many researches e.g. Tucker (2013), Kumar et al. (2016) and Vlahostergios (2018).

In the presented study a more advanced methodology approach towards the prediction of the flow around the quadcopter propeller is adopted regarding turbulence modelling. The selected applied turbulence models follow three different turbulence modelling philosophies. The first model is the linear eddy-viscosity SST k-ω of Menter (1994), which is used as a reference base and it is the most widely used turbulence model for this kind of aerodynamic studies. The other two more advanced models are the baseline Reynolds stress model BSL-RSM (Ansys CFX theory guide 2009) and the explicit algebraic model EARSM which is based on the model of Wallin and Johansson (2000).

## 2. Propeller geometry and CFD modelling details

As an initial propeller geometry configuration, a NACA airfoil-based propeller design of D=35.6 cm diameter was selected. For the CFD computations and the analysis of the computational results, the ANSYS-CFX (Academic Student) software was used. An indicative view of the propeller geometry and the computational domain with the imposed boundary conditions can be seen in Figure 1. The imposed boundary conditions were selected based on the drone mission and are provided in Table 1.

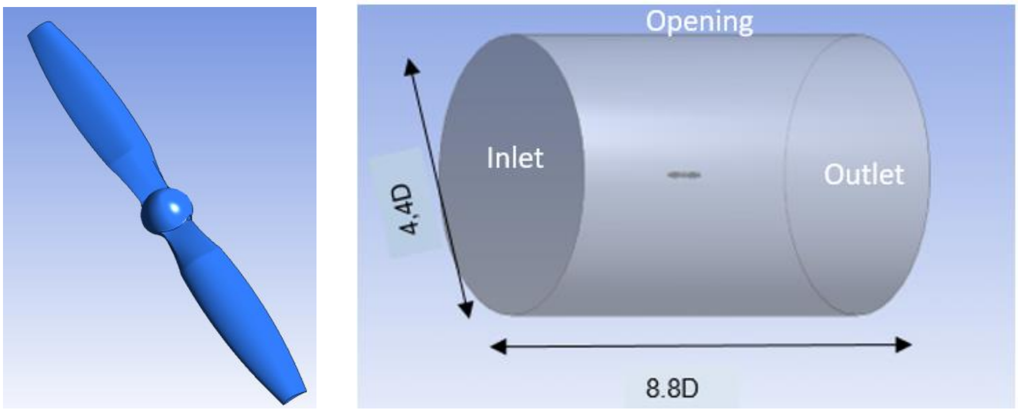


*Figure 1: Indicative view of the propeller geometry and the computational domain*

*Table 1: Implemented Inlet Boundary conditions*

| Inlet (Flight) velocity | Turbulent intensity | Propeller Angular velocity |
|---|---|---|
| 8m/sec | 1% | 10,000 rpm |

For the turbulence modelling, three different turbulence models were selected each one based on a different turbulence modelling approach. More specifically, the models were the low-Reynolds shear stress transport (k-ω SST) linear eddy viscosity model of Menter (1994). This model is widely used by industries and engineers since it provides stable convergence and uses a blending function for combining the benefits of turbulent dissipation (ε) away from the walls and the specific turbulence dissipation (ω) near the wall regions for modelling the turbulent destruction. The k-ω SST transport equations are provided by Eq(1-2).

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho U_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_\tau}{\sigma_\kappa}\right)\frac{\partial k}{\partial x_j}\right] + P_k - \beta' \rho k \omega \tag{1}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho U_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_\tau}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 \tag{2}$$

The second model is a Reynolds stress turbulence model (RSM), named as the baseline RSM (BSL-RSM) as described in Ansys CFX theory guide (2009). The BSL-RSM adopts also the same blending function approach as the previous model, for modelling the turbulent destruction near the walls, and uses the Reynolds stress transport equations for modelling turbulence, as given by Eq(3).

$$\frac{\partial\left(\rho\overline{u_i u_j}\right)}{\partial t}+\frac{\partial\left(\rho U_\kappa\overline{u_i u_j}\right)}{\partial x_\kappa}=\frac{\partial}{\partial x_\kappa}\left[\left(\mu+\frac{2}{3}C_s\rho\frac{k^2}{\varepsilon}\right)\frac{\partial\overline{u_i u_j}}{\partial x_\kappa}\right]+P_{ij}-\frac{2}{3}\delta_{ij}\rho\varepsilon+\Phi_{ij} \quad (3)$$

Finally, the third selected model is an Explicit Algebraic Reynolds stress model (EARSM) based on the model proposed by Wallin and Johansson (2000) which solves a linear set of algebraic equations for computing the Reynolds Stresses and uses the transport equation of ε for modelling the turbulent destruction process. Additional information regarding the adopted models, the processes of turbulent generation $P_k$, pressure strain correlation $\Phi_{ij}$ and the constants of the provided equations can be found in Ansys CFX theory guide (2009).

For simulating the propeller rotation in relation to the drone flight velocity, all the transport equations were solved in a rotating frame of reference. This methodology which was also followed by other researchers, Salpingidou et al. (2018), adds additional source terms in the RANS momentum equations in order to take into consideration the Coriolis and the centrifugal forces that are required for modelling the momentum that is added in the flow by the propeller rotation. The additional source terms are given by Eq(5-6).

$$S_{cor}=-2\Omega\times U \quad (5)$$

$$S_{cfg}=-2\Omega\times(\Omega\times R) \quad (6)$$

Where Ω is the angular velocity of the propeller and U in the velocity corresponding to the flight velocity of the drone, related to the absolute reference frame. The implemented values of these quantities are provided in table 1.

In order to solve the momentum and turbulent transport equations, a grid of ~500K computational cells was constructed. Special care was taken near the propeller wall regions mesh where $y^+$ was targeted to be less than unity for all the turbulence models in order to accurately resolve the boundary layer development. Representative views of the computational mesh around the propeller can be seen in Figure 2.

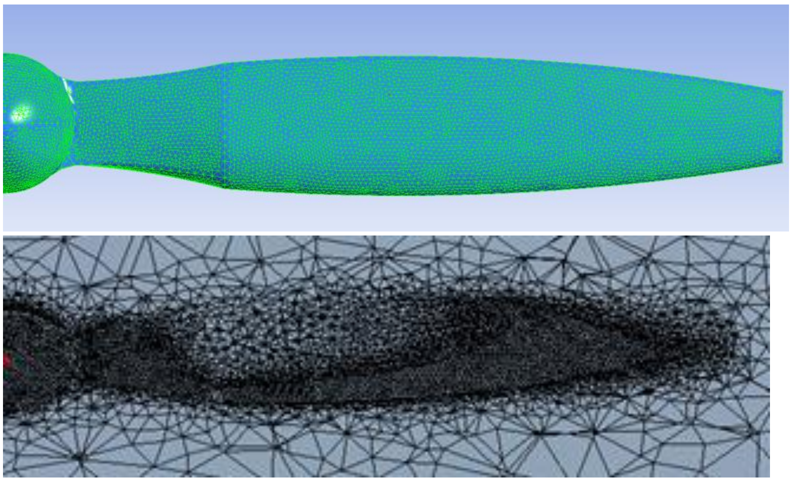

*Figure 2: Representative views of the computational mesh around the propeller*

For all the transport equations (momentum and turbulent) a high order resolution scheme was selected in order to provide more accurate results and it is the one described by Barth and Jespersen (1989). The convergence for all the models was terminated when the residuals of all the quantities reached values less than $10^{-5}$.

## 3. Results and discussion

For providing a clear view regarding the propeller aerodynamic behavior and have a preliminary assessment of the propeller performance, the pressure distributions were examined for all turbulence models. Five representative stations along the spanwise direction of the propeller were selected, which can be seen in Figure 3.

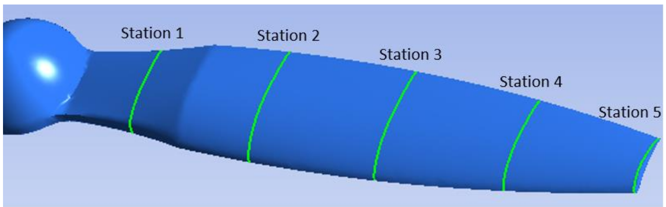


*Figure 3: Selected stations along the propeller spanwise direction*

The selected locations of each station along the propeller measured from the spinner centre are provided in table 2.

*Table 2: Presented locations along the propeller*

| Station 1 | Station 2 | Station 3 | Station 4 | Station 5 |
|---|---|---|---|---|
| 3.6 cm | 7.1 cm | 11 cm | 14.2 cm | 17.7 cm |

The pressure coefficient (Cp) distributions of the 5 stations can be seen in Figure 4 along the Z-axis (streamwise direction) normalized with the chord length of the airfoil of each station.

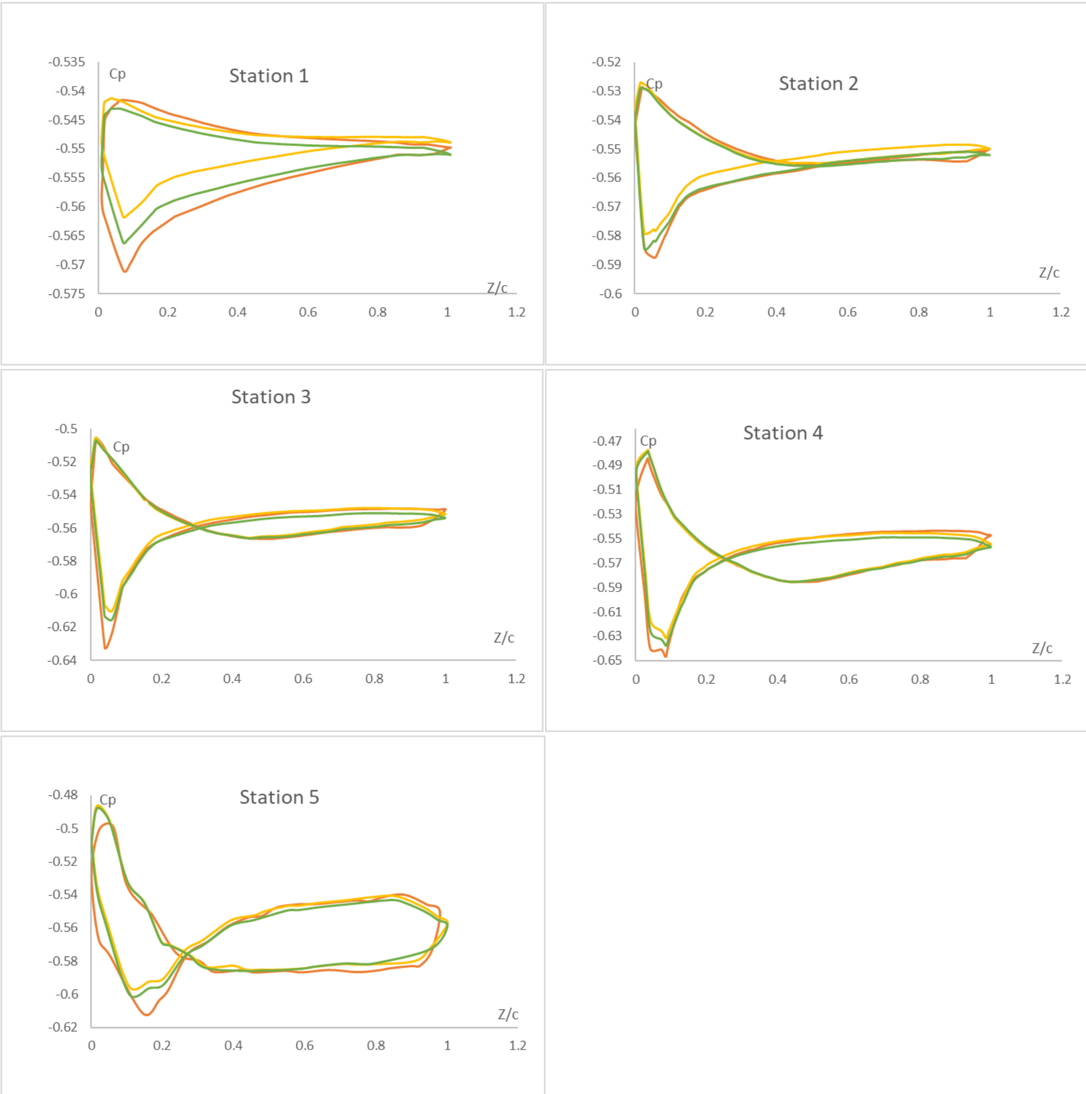


*Figure 4: Cp distributions along the propeller selected stations (lines: red-EARSM, yellow-SST, green BSL-RSM)*

From Figure 4 it can be concluded that the selected propeller in the original adopted design is not suitable for integration on the drone without any further modifications for the current flight conditions. The Cp distributions do not show beneficial pressure distributions around the propeller in order to be used with its original geometry. Regarding the turbulence models capabilities in calculating the correct Cp, no significant differences can be observed. The BSL-RSM and the SST k-ω models provide similar distributions in comparison to the EARSM, which show a slightly deviating behavior of the pressure distributions, due to the same modelling methodology that they adopt. Additional investigations are needed in order to have a more in-depth view of the impact of the different modelling terms of each turbulence model to the aerodynamic study and the propeller efficiency. Figure 5 provides indicative contour plots of the pressure distributions for stations 2 and 3 which are the station located closer to the mid span of the propeller blade.

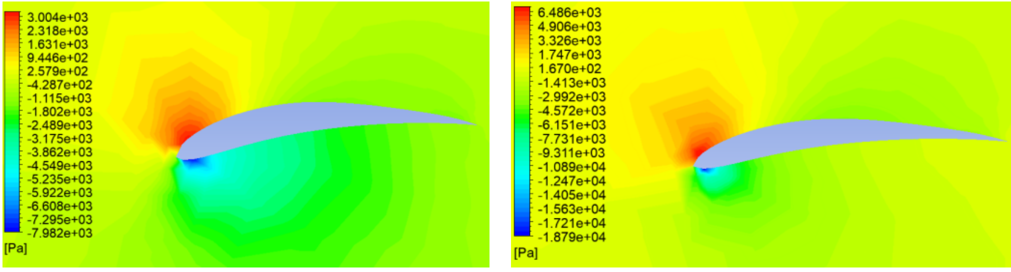


*Figure 5: Pressure contours for stations 2(left) and 3(right)*

The presented figures indicate that a propeller redesign is necessary in order to have the desired pitch angle in all stations. As a result, an improved propeller design is needed in order to provide the maximum aerodynamic efficiency and thrust in a wider range of inlet conditions. Figure 6 presents a propeller view with representative pressure contours around the propeller walls being aligned with the previous conclusions. It is evident that a twist of the propeller is required in order the stagnation point to be as much as possible closer to the leading edge of the propeller and thus, increase its efficiency.

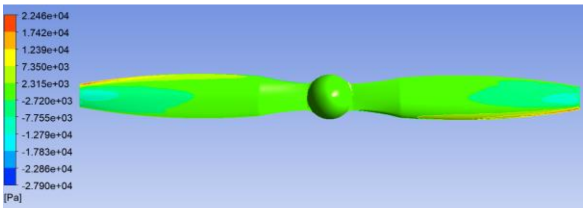


*Figure 6: Pressure contours and Turbulence intensity distribution on around the propeller*

In order to examine the turbulence models behavior, contours of turbulence intensity (Tu) are presented in Figure 7. Additionally, in the same figure, streamlines of the flow for the rotating reference frame are also presented regarding the modelling of the rotating reference frame.

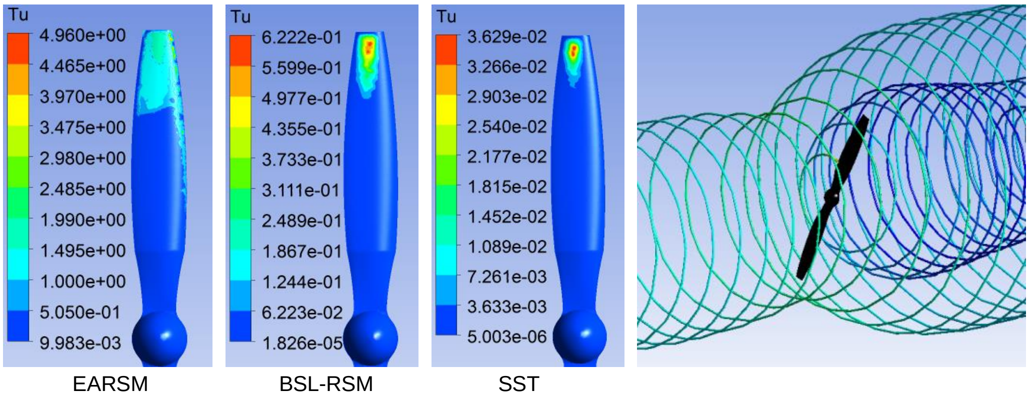


*Figure 7: Tu distributions (left) and flow streamlines for the rotating reference frame (right)*

As it can be seen from Figure 7, the ω-based turbulence models (BSL-RSM and SST) present similar behaviour. They provide the maximum Tu in the region where maximum turbulence generation is expected, which is also in line with the Cp calculation results. However, the SST model computes reduced Tu in comparison to the BSL-RSM. This is probably related to the dumping of the eddy viscosity near the wall regions that the model implements which is not present in the BSL-RSM. On the other hand, the EARSM predicts more diffused and much higher Tu values near the propeller tip. This behaviour is linked to the ε transport equation adoption, for modelling the turbulence destruction, which is also adopted in the vicinity of the propeller walls.

## 4. Conclusions

In the current work, a performance assessment of an initially selected propeller for a quadcopter drone is presented, with the use of various turbulence models that follow different modelling approaches. This is the preliminary work of a bigger project that aims in the designing and manufacturing of a quadcopter drone for participating in the UAS Challenge competition, which will be materialized by the DART student team of Democritus University of Thrace. Aspects as energy saving, flight efficiency, cost manufacturing minimization and use of environmentally friendly materials are some of the main targets of the project. The quadcopter will use electrical energy and will be entirely 3D printed with recyclable material. For the study, the turbulence models were assessed for a preliminary quantification of the drone propeller aerodynamic behavior in conditions that will be met in the competition. The SST k-ω linear eddy viscosity model, the BSL-RSM model and an Explicit Algebraic turbulence model were adopted. The different modelling approaches were selected since there is no availability of experimental data for the specific design. As a result, it was considered a good approach to proceed with calculations with turbulence models that adopt different modelling philosophies. Based on the authors knowledge, there are no studies that apply advanced turbulence models for assessing their capabilities in the aerodynamic design of quadcopters and their components. The study showed that there was no significant difference between the results that the turbulence models provided regarding the propeller aerodynamic behavior and the pressure distributions around the propeller. However, the use of the BSL-RSM model seems a reasonable choice to proceed since it can compute the Reynolds stress anisotropy and provide better predictions for transitional and detaching boundary layers which are met in complex propeller flows. The path towards the propeller geometry design improvement was indicated by the pressure distributions to various selected stations along its spanwise direction. Valuable information was gathered for improving the propeller pitch angle and its positioning to the flow for maximum efficiency. For the final drone layout, more studies are needed for designing a propeller with optimized performance for a wider range of aerodynamic conditions.

### References

ANSYS CFX-Solver Theory Guide, 2009, Release 12.1, ANSYS Inc., Canonsburg, PA, USA.

Barth, T.J., Jesperson, D.C., 1989, The Design and Application of Upwind Schemes on Unstructured Meshes. In Proceedings of the 27th Aerospace Sciences Meeting, Reno, NV, USA, 9–12.

Sarghini F., De Vivo A., 2017, Analysis of Preliminary Design Requirements of a Heavy Lift Multirotor Drone for Agricultural Use. Chemical Engineering Transactions, Volume 58.

Kumar, G.S., Azad, D., Mohan, K.L., 2016, Evaluation of turbulence modelling on low speed centrifugal compressor using low speed centrifugal compressor using computational fluid dynamics. International Journal of Mechanical Engineering and Technology, Volume 7, 634–641.

Kutty, H.A., Rajendran P., 2017, 3D CFD Simulation and Experimental Validation of Small APC Slow Flyer Propeller Blade. Aerospace, 4(1), 10.

Menter, F. R., 1994, Two-equationEddy-viscosity Turbulence Models for Engineering Applications. AIAA Journal, 32 (8), 1598–1605.

Salpigidou, C., Misirlis, D., Vlahostergios, Z., Yakinthos, K., 2018, Development of surrogate models for the prediction of the flow around an aircraft propeller. International Journal of Sustainable Energy, Volume 37, Issue 5, 498-514.

Tucker, P.G., 2013, Trends in turbomachinery turbulence treatments. Progress in Aerospace Sciences, Volume 63, 1-32.

Vlahostergios, Z., 2018, Performance Assessment of Reynolds Stress and Eddy Viscosity Models on a Transitional DCA Compressor Blade. Aerospace, Volume 5, 102.

UAS-Challenge., 2019, Institution of Mechanical Engineers, <www.imeche.org/events/challenges/uas-challenge/about-uas-challenge> accessed 01.04.2019.

Wallin, S., Johansson A., 2000, A complete explicit algebraic Reynolds stress model for incompressible and compressible flows, Journal of Fluid Mechanics, 403, 89-132.

Yeong S. P. and Dol S. S., 2016, Aerodynamic Optimization of Micro Aerial Vehicle. Journal of Applied Fluid Mechanics, Volume 9, No. 5, 2111-2121.