Journal of Aeronautics and Space Technologies
15(2) (2022) 1-12
Journal Homepage: jast.hho.msu.edu.tr

RESEARCH ARTICLE

An Open-Source Aerodynamic Shape Optimization Application for an Unmanned Aerial Vehicle (UAV) Propeller

Bir İnsansız Hava Aracı Pervanesi İçin Açık Kaynaklı Bir Aerodinamik Şekil Optimizasyon Uygulaması

Şahin Yiğit[1*], Saleh Abuhanieh[1], Barış Biçer[1]

[1] *Turkish Aerospace, OpenSource CFD Group, 06980 Ankara, Turkey,*

Received: *August 17, 2021* ***Revised:*** *January 20, 2022* ***Accepted:*** *April 06, 2022*

Abstract
The Unmanned Aerial Vehicles (UAVs) have become more popular and functional for today's needs in comparison to the conventional aircrafts for military, agriculture and private purposes. Generally, these vehicles are using propeller based propulsion systems. Thus, aerodynamic shape optimization is a vital process to improve their performance. Accordingly, in the present study, a Computational Fluid Dynamics (CFD) based open-source shape optimization framework (combined use of a CFD solver, Optimization solver, Free-Form Deformation (FFD) tool and Mesh Generator utility) is designed to optimize the shape of a generic UAV propeller. As a result, the open-source shape optimization framework here woks quite efficiently and the Figure of Merit (FM) of the propeller has been improved by around 40 %.

Keywords: Open-source, Aerodynamic shape optimization, UAV, Propeller, Gradient free optimization, Gradient-based optimization

Özet
Askeri, tarım ve özel amaçla kullanılan insansız hava araçları günümüz şartlarında insanlı hava araçlarına göre daha popüler ve fonksiyonel olmaktadır. İnsansız hava araçları genellikle pervane tahrikli bir itki sistemi kullanmaktadır. Bu sistemlerin performanslarının iyileştirilmesi için aerodinamik şekil optimizasyonu hayati önem taşımaktadır. Bu nedenle bu çalışmada, bir açık kaynaklı, Hesaplamalı Akışkanlar Dinamiği (HAD) tabanlı bir optimizasyon döngüsü (HAD çözücüsü, optimizasyon çözücüsü, serbest şekil değiştirme aracı ve çözüm ağı üreticisi birlikte çalışmakta) tasarlanarak, bir generik insansız hava aracı pervanesi üzerinde test edilmiştir. Tasarlanan bu açık kaynaklı şekil optimizasyonu döngüsünün oldukça etkili çalıştığı görülmüş ve sonucunda söz konusu pervanenin yararlılık katsayısı % 40 iyileştirilmiştir.

Anahtar Kelimeler: Açık kaynak, Aerodinamik şekil optimizasyonu, İnsansız hava aracı, Pervane, Genetic algoritma, Gradyan tabanlı optimizasyon

1. INTRODUCTION

The majority of unmanned aerial vehicles (UAVs) use a propeller-based propulsion system. In the design stage of such systems, it is vital to satisfy aerospace industry´s goals such as high fuel-efficiency, long-endurance and emission reduction. This can be achieved by improving the design efficiency using robust aerodynamic shape optimization (ASO) framework. However, the ASO requires a lot of testing iteration on the real prototype which is resources and time-consuming process even for a single propeller. Therefore, the numerical based ASO has become an efficient tool for many different applications [1-4]

*Corresponding Author
E-mail: sahin.yigit@tai.com.tr

for engineers from different disciplines in the last decade due to the development of computers, efficient optimization algorithms and numerical methods.

Some numerical methods such as Blade Element Method (BEM) [5, 6] and panel method (PM) [7] have been used to model the rotating flow of propellers. It has been reported that both BEM and PM are computationally effective and their results agree with experimental results when flow is subsonic. Therefore, these methods can be used in the ASO studies for rapid-design of propellers [5-8] for low Mach number flows. However, the aforementioned methods cannot account for flow separation on the blades or other effects such as shocks (for compressible cases), tip vortices or hub horseshoe vortices. It is worth noting that there are applications where propeller works in the transonic flow regime although the flow conditions of aircraft is subsonic. Therefore, there is also necessity of high-fidelity ASO tools where Navier-Stokes solution [9-12] is required for the final design of such propellers. It is obvious that Navier-Stokes solver based ASO is more expensive in comparison to the low-fidelity methods (e.g. BEM and PM). However, even small improvement (for instance in Figure of Merit of such propellers which works in the transonic flow regime) provides considerable enhancement in terms of fuel consumption. Therefore, Navier-Stokes based CFD, ASO has been utilized beneficially for different purposes such as improving force distribution on the propeller and aerodynamic characteristics of propeller (e.g. Figure of Merit, minimizing torque level for a given required thrust) [13-15].

There are several commercial Navier-Stokes based CFD ASO tools which have being already used in the aerospace industry in order to reduce cost and time for the wind tunnel testing [2,16]. However, the commercial softwares are generally quite expensive in terms of license costs and it limits the users in the black box sense. Therefore, the present work is directed to offer a practical open source Navier-Stokes based CFD ASO framework tailored for the aerospace industry. It should be noted that the novelty of the current work is that the CFD solver and the mesh generator stages of the proposed ASO framework can be replaced with computationally cheaper alternatives (e.g. BEM or panel method for subsonic flow) in order to redesign the current cycle as a low-fidelity open-source ASO framework. Therefore, the main purpose here, to offer a general open-source shape optimization for both low and high fidelity shape optimization applications. Accordingly, the objectives of the current study are:

i. To design a general open-source ASO framework where there is no license requirements.
ii. To demonstrate each component of the aforementioned open-source ASO framework and how they work.
iii. To test the proposed strategy in shape optimization by using a simple test case (i.e. a generic UAV propeller).

2. METHODOLOGY

Figure 1 shows the work-flow of the optimization cycle proposed in the present work which consists of OpenFOAM (open-source CFD solver) [17], Dakota (open-source optimization tool) [18], MiMMO (open source C++ library for manipulation and morphing of surface and volume meshes) [19] and SnappyHexMesh (open-source volume mesh generator) [17].

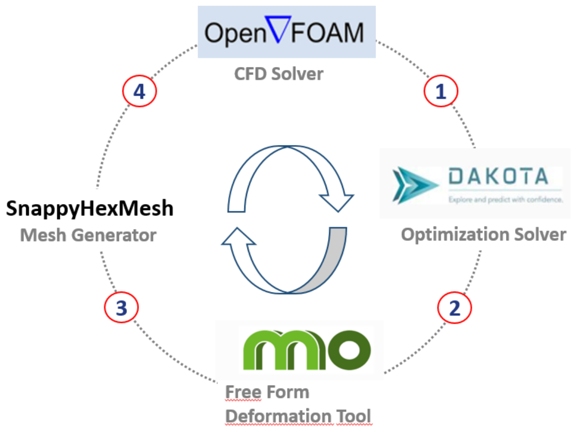

Figure 1. Schematic diagram of the proposed open-source CFD-based ASO framework.

In Fig. 1, step 1 exhibits numerical modelling of rotating flow with CFD solver for a generic propeller, where its properties are given in Table 1. Step 2 shows, optimization solver, which provides new shape parameters based on object function, geometric constraints and optimization algorithm. In step 3, the free-form deformation tool deforms the surface mesh (i.e. gives a new stl file) based on the displacement information provided by the optimization solver. In step 4, the mesh generator creates new volume mesh for the new fluid domain for the CFD solver. This cycle can be continued until certain level of enhancement is achieved in the efficiency of the propeller. It is worth mentioning here that the CFD solution has been obtained from no-flow condition for each evaluations, previous evaluation has not been interpolated as an initial for the upcoming one.

Table 1. View of the generic propeller with its properties.

Number of Blades	2
Rotor Radius (R)	0.1905 m
Rotor Blade Mean Chord (c)	0.03153 m
Aspect Ratio (R/c)	6.0418
Rotor Solidity (s)	0.10411

Table 1 demonstrates the utilized generic UAV propeller and its properties. This propeller has been used as an example application in the current study due to its experimental data availability for validation. The thrust values obtained from the present work are compared with the experimental results in Ref. [20].

2.1. Step-1 (Numerical Modelling of the Rotating Flow for a Generic UAV Propeller)

The Single Reference Frame (SRF) methodology has been used for modelling the rotating flow problem in the current analysis. The steady-state incompressible mass and momentum equations (Eqs. 1 and 2) have been solved in conjunction with the boundary conditions which are illustrated in Fig 2.

$$\nabla.(u_r) = 0 \tag{1}$$

$$\nabla.(u_r u_r) = -\nabla p + \nabla.(v \nabla u_r) - 2\Omega \times u_r - \Omega \times \Omega \times R \tag{2}$$

where p, ρ, μ, R denote pressure, density, dynamic viscosity and the propeller radius, respectively. The relative velocity is defined as $u_r = u - \Omega \times R$, where Ω is the angular velocity.

The computational domain is modelled with single blade of the propeller by using periodic Arbitrary Mesh Interface (AMI) (i.e. cyclicAMI) as shown in Fig 2. To solve the non-linear set of governing equations in finite volume framework, the open-source CFD package OpenFOAM has been utilized. The pressure-velocity coupling has been addressed by using the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm. For the discretization of the convective term, a first-order upwind scheme is used, while for the diffusive terms, a second-order central difference has been utilized.

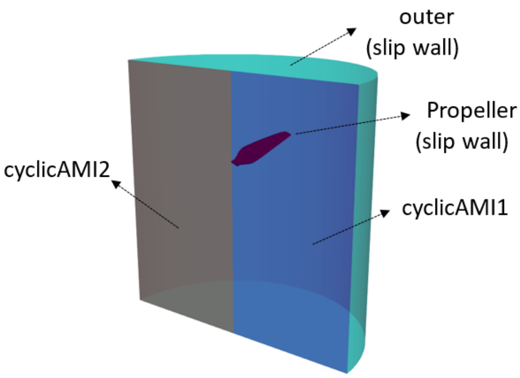

Figure 2. Geometric view and boundary conditions of the flow domain.

It is worth noting that, in this work, the standard OpenFoam solver simpleFoam has been used along with the MRFProperties dictionary. However, to run the case as SRF, the whole domain has been defined as a rotating zone while keeping the cyclic patches as nonRotatingPatches in the MRFProperties file.

An Euler mesh configuration with around 750 thousand cells is generated as shown in Fig. 3 using the open-source meshing tool snappyHexMesh. The numerical results from the current analysis have been validated using the experimental results reported by Soydan et al. [20]. They have experimentally investigated a co-axial rotor performance in terms of thrust for the same propeller in the hover condition. The relative error between numerical and experimental [20] results in the thrust values has been found to be less than 4 % as shown in Table 2. It should be noted that the full domain with two blades have been considered in the validation using a second-order upwind discretization scheme for the convection term.

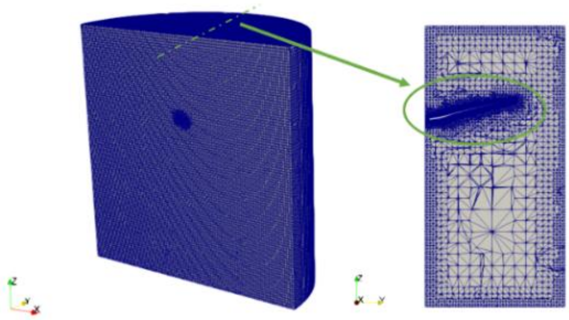

Figure 3. Unstructured Euler mesh for the propeller (around 750 thousand cells).

Table 2. Validation of the numerical findings with experimental results [20].

Throttle (100 %)	Angular velocity (rad/s)	Exp. Max. Tip Mach Number (Ma_{tip})	Exp. Thrust (kg)	Num. Thrust (kg)
5980 rpm	626.22	0.3506	1.740	1.801

Additionally, a finer Euler mesh is also generated to check the effects of the mesh size on the thrust value. The fine mesh contains around 7.5 Million cells which is about 10 times the original/coarse mesh size. The difference in the thrust value has been found to be less than 1 % for the finer mesh configuration. Thus, the Euler mesh configuration which is given in Fig. 3 has been used in the optimization cycle throughout this study for the sake of reducing the computational cost. Additionally, the same configuration is also modelled by using compressible solver (i.e. rhoSimpleFoam) and it has been found that the incompressible and compressible solutions are in quite good agreement for the current application.

2.2. Step-2 (Optimization with DAKOTA)

Optimization process aims to minimize (or maximize) a single objective function (or multiple functions) subject to constraints on the design variables. It is worth noting that the objective function, which is calculated by the CFD solver, has been chosen here as the Figure of Merit (FM). FM measures the hover performance that is key to the efficiency of rotor system. FM can be defined as:

$$FM = {C_T}^{1.5} / (\sqrt{2} C_Q) \tag{3}$$

where $C_T = T/\rho(\Omega R)^2 \pi R^2$ and $C_Q = Q/\rho(\Omega R)^2 \pi R^3$ are the thrust and torque coefficients, respectively. Here, ρ, T, Q represents density, propeller thrust (N) and propeller torque (N.m), respectively.

Accordingly, the FM of the propeller is aimed to be maximized at 5980 rpm rotational speed (direction of rotation is $+z$) in the hover condition with the geometric constraints

($-0.004 < x(m) < 0.004$, $-0.001 < z\ (m) < 0.001$, no change in the y direction (i.e.R)) by using the optimization tool DAKOTA [18].

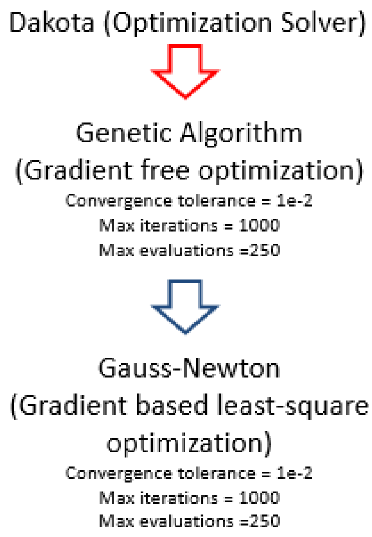

Figure 4. Schematic of the hybrid sequential optimization method for one iteration.

DAKOTA is an open-source black-box optimization solver which has been successfully used in earlier CFD-based ASO studies [21,22]. It offers many optimization schemes, which include gradient and non-gradient based optimization algorithms. Gradient-based optimization algorithms (e.g. Gauss-Newton solvers) provide efficient performance for smooth and continuous variables and converge fast to the closest minimum near initial. However, the Gradient-based optimization algorithms do not give guarantee for global minimum convergence and they are computationally expensive. The most expensive part is the gradient calculation, since for each iteration of the optimization process, the gradient is calculated using finite difference for each of the design variables separately. i.e. to calculate the gradient for N variables one needs to evaluate the objective function (the CFD solver in our case) at least N times. That makes the whole process immensely expensive. Unlike using the finite difference for evaluating the gradient, the adjoint approach is preferred when the number of design variables are large. With the adjoint approach, the gradient for all the design variables are calculated within the same objective function evaluation iteration. However, the adjoint approach can be only implemented within the flow solver and can't be used within the black-box optimization tools.

Thus, gradient-free based optimization algorithms (e.g. genetic algorithm) can be used to search global minimum of the object function more efficiently specially while using a black-box optimization tool.

According to the above discussion, a hybrid sequential methodology has been used to apply more efficient optimization. In the hybrid method, the goal is to use the strengths of different optimization methods as sequential minimization (or maximization) process. In the current analysis, the Genetic Algorithm (GA) is used to produce better initial conditions for Gauss-Newton Gradient-based optimization method as it is shown in Fig. 4.

The theory of the optimization methods are not discussed here as DAKOTA [18] is used here in the black-box sense. Interested readers are referred to Ref. [18] for extensive details of the optimization algorithms and their implementations.

2.3. Step-3 (Free-Form Deformation with Mimmo)

In the present study, a Free-Form Deformation (FFD) tool (MiMMO library [19]) has been used to generate the new shape at each iteration. For MiMMO, the FFD box is defined by series of control points as shown in Fig. 5. These control points are relocated continuously based on some polynomial or radial basis functions in the FFD box to generate smooth displacement for morphing the shape.

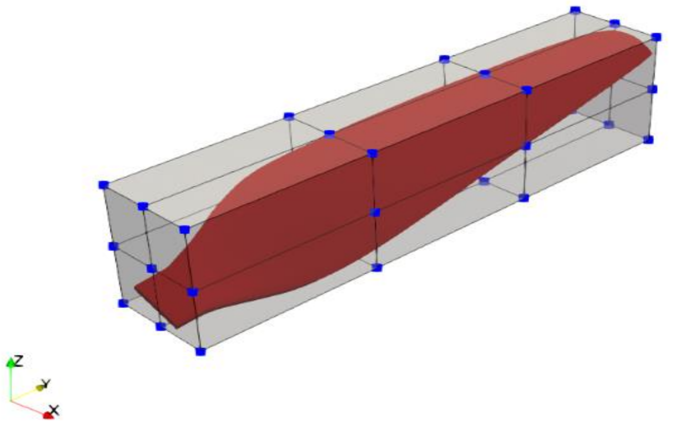

Figure 5. FFD box with deformable control points (blue points) with un-deformed geometry of the propeller inside the FFD box.

As it can be seen in Fig. 5, the FFD box is defined with 36 number of control points in the present study. Since the given geometric constraints (i.e. ($-0.004 < x(m) < 0.004$, $-0.001 < z\ (m) < 0.001$, no change in the y direction (i.e.R))) in order to keep the length of propeller constant, these control points can only move in the x and z directions which brings 72 number of design variables for DAKOTA to optimize based on the object function (maximizing FM of the propeller). Finally, MiMMO deforms the propeller`s surface mesh (i.e. gives a new stl file) based on the information provided by DAKOTA.

2.4. Step-4 (Generating Volume Mesh with SnappyHexMesh)

The open-source mesh generation utility SnappyHexMesh [17] has been utilized in here to generate a volume mesh for each iteration of the optimization cycle as given in Fig. 1. It is worth noting that MiMMO deforms only the surface mesh but not the volume mesh of the domain in the current analysis. This is because, there is a chance to create negative cells in the domain while deforming the volume mesh. Thus, for each iteration/evaluation of the optimization cycle in Fig. 1, a new volume mesh is produced to avoid any negative cells in the domain which cause the optimization cycle to stop.

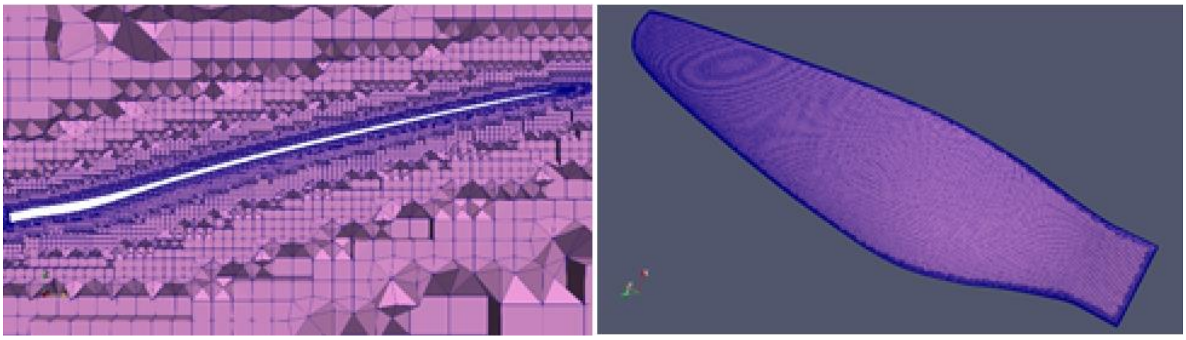

Figure 6. Surface mesh of the propeller (top) and slice from the volume mesh (bottom) which is used in the optimization.

Fig. 6 shows the propeller surface mesh (top) and slice from the volume mesh (bottom) which is generated by using snappyHexMesh with maximum refinement level of 5 for the background mesh (2×2×2 m with 11 cells in each direction). This mesh contains 750 thousand cells and it is used in the optimization cycle.

3. RESULTS AND DISCUSSION

Fig. 7 shows the change of the object function for each evaluation of the CFD-based optimization cycle proposed here. It can be easily seen from Fig. 7 that the object function (i.e. FM) increases with increasing number of evaluations. It is worth to mention that the optimization cycle has been limited to 250 evaluations in the present application. The number of evaluations can be increased naturally however, it is suggested to stop optimization cycle when the required improvement has been achieved in order to avoid unnecessary computational cost and excessive data storage.

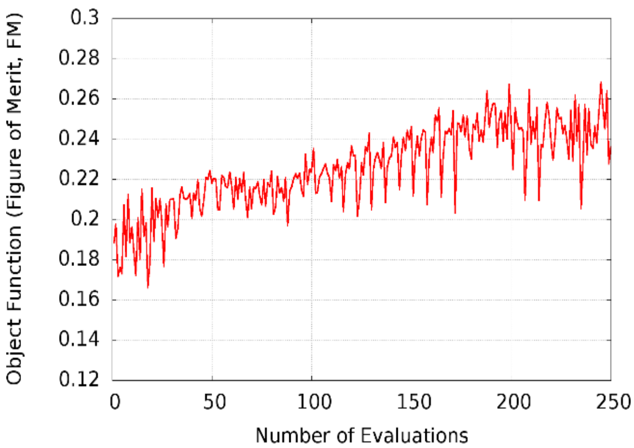

Figure 7. The object function (i.e. FM) versus number of evaluations.

As a result of the aforementioned optimization cycle for 250 evaluations, the object function (i.e. FM) has been maximised by around 40 % relatively as it is shown in Table 3. Here, it should be noted that the rotor solidity has changed around 5 % after optimization. This can also affect results of FM slightly.

Table 3. The results of the CFD-based open-source optimization cycle proposed in this study.

Object Function	Before Optimization	After Optimization	Relative Improvement (%)
Figure of Merit (FM)	0.188	0.268	42

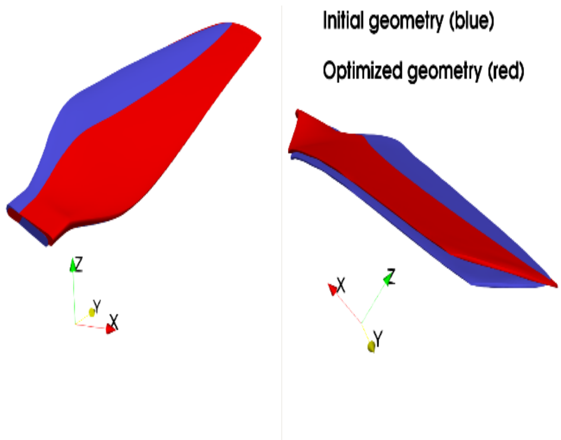

Figure 8. The view of initial and optimized geometries of the propeller from different views.

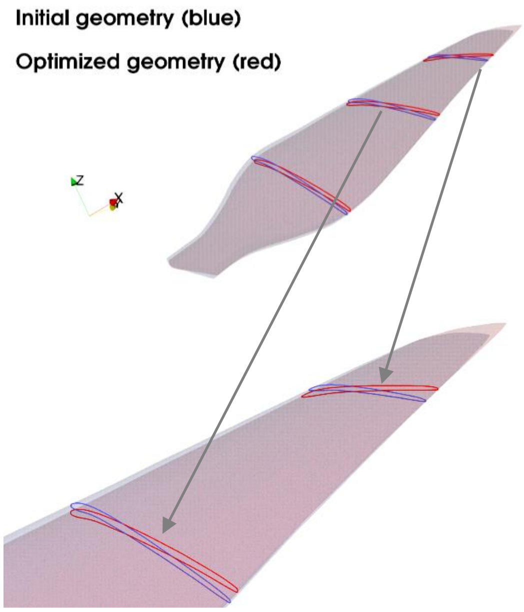

Figure 9. A view of the initial and optimized geometries of the propeller (different slices along y-axis).

Figs. 8 and 9 show the differences between initial and the optimized geometries of the propeller. It is noticeable from Fig. 8 that the optimized geometry is larger on the x direction and the collective angle of the propeller is optimized (Fig. 9) in order to maximize hover performance.

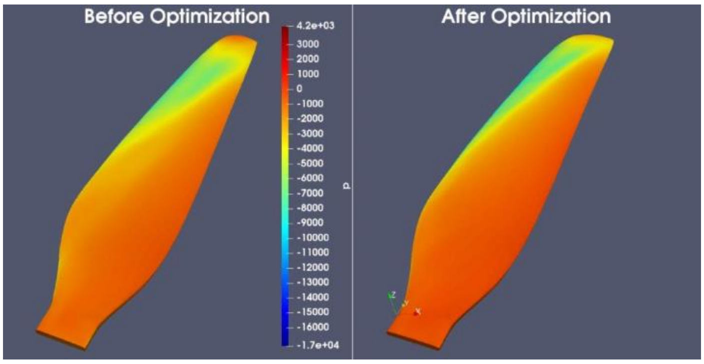

Figure 10. The pressure distributions on the propeller.

Fig. 10 shows the pressure distribution over the propeller before and after optimization, however it should also be noted from Fig. 9 that the collective angle of the propeller is also changed after the optimization.

4. CONCLUSION

In the present study, rotating flow of an unmanned aerial vehicle (UAV) propeller has been modelled numerically and validated by an experimental results. This numerical model has been used to design a CFD-based open-source shape optimization framework to enhance the hover performance of the propeller. Accordingly, CFD based open-source shape optimization framework (combined use of CFD solver, Optimization solver, Free-Form Deformation tool and Mesh Generator utility) has been tested on the aforementioned propeller. The Figure of Merit (FM) of the propeller has been improved around 40 % by using the proposed open-source shape optimization framework in this work. Overall, the results show that the proposed CFD-based open-source shape optimization framework works promisingly well, free-license and quite easy to operate for application engineers. It is believed that this methodology can be utilized efficiently for many different applications in aerospace as well as in other industries.

REFERENCES

[1] B. Mohammadi, O. Pironneau, Applied shape optimization for fluids, Oxford University Press, 2009.

[2] C. Lui, B. Weiyang, X. Dong, *Multi-objective shape optimization of a plate-fin heat exchanger using CFD and multi-objective genetic algorithm*, Int. Journal of Heat and Mass Transfer **111**, 65-82 (2017).

[3] H. Kim C.A. Yang, *A new surface modification approach for CFD-based hull form optimization*, Journal of Hydrodynamics **22**, 503-508 (2010).

[4] M. Horvat, L. Bruno, S. Khris, L. Raffaele, *Aerodynamic shape optimization of barriers for windblown sand mitigation using CFD analysis*, Journal of Wind Engineering & Industrial Aerodynamics **197**, 104058 (2020).

[5] J. Sodja, D. Stadler, T. Kosel, *Computational Fluid Dynamics Analysis of an optimized load-distribution propeller*, Journal of Aircraft **49**(3), 955-961 (2012).

[6] J. D. Hoyos, J. H. Jimenez and P. Alvarado, "Improvement of electric aircraft endurance through propeller optimization via BEM-CFD methodology," *2nd Int. Conf. On Trends in Mechanical and Aerospace*, *Washington, United States*, *September 25-27, 2020*.

[7] J. L. Hess and W.O. Valarezo, *Calculation of steady flow about propeller using a surface panel method*, Journal of Propulsion **1**(6), 470-476 (1985).

[8] C.C. Smith, N. R. Garcia, O. Sigmund and C. S. Andreasen, *Aerodynamic shape optimization of aircraft wings using panel method,* AIAA Journal **58**(9), 3765-3776 (2020).

[9] N. Ashton, V. Skaperdas, *Verification and validation of OpenFOAM for high-lift aircraft flows*, Journal of Aircraft **56**(4), 1641-1657 (2019).

[10] D. Sandeep, Y.P. Ravitej, S. Khot, R Ravikumar, Abhinandan, N. Kumar, S.B. Karthik and Veerachari, ''CFD simulation of transonic flow past NACA 0012 aerofoil,'' *In AIP Conference Proceedings 2316(1)*, pp. 020013 (2021).

[11] S. Abuhanieh, B. Bicer, M. Sahin, "A validation for the OPENFOAM-HISA solver for drag prediction" *32nd Int. Conf. on Parallel CFD*, *Nice, France, May 17-19, 2021*.

[12] B. Bicer, C.A.S. Metin, B. Celik, "An Investigation on shock wave boundary layer interactions over awing via OpenFOAM" *7th OpenFOAM Conference*, *Duisburg, Germany, July 23-26, 2019*.

[13] G. Droandi, G. Gibertini, *Aerodynamic shape optimisation of a proprotor and its validation by means of CFD and experiments*, The Aeronautical Journal **119**, 1223-1251 (2015).

[14] A.V. Lysenkov, "The propeller optimization with the use of CFD" *Int. Conf. On the Methods of Aerophysical Research, Novosibirsk, Russia, August 13-19, 2018*.

[15] H.I. Kwon, S. Yi and S. Choi, *Design of efficient propellers using variable-fidelity aerodynamic analysis and multilevel optimization*, Journal of Propolsion and Power **31**(4),1057-1072 (2015).

[16] J. Morgado, M. Abdollahzadeh, M.A.R. Silvestre, J.C. Pascoa, *High altitude propeller design and analysis*, Aerospace Science and Technology **45**, 398-407 (2015).

[17] "OpenFOAM Web Page," [Online]. Available: http:// openfoam.com. [Accessed: August 11 2021].

[18] "DAKOTA Web Page," [Online]. Available: https:// dakota.sandia.gov/ [Accessed: August, 11, 2021].

[19] "MiMMO Web Page," [Online]. Available: https:// optimad.github.io/mimmo. [Accessed: August, 11, 2021].

[20] A. Soydan, H. Sahin, B. Bicer, S. Sariozkan, M. Sahin, "Experimental and Numerical Investigation of Co-Axial Rotor Interaction to Thrust," Progress in Computational Fluid Dynamics, An Int. Journal, 2021 (in-press).

[21] C.C. Xia, Y.J. Gou, S.H. Li, W.F. Chen and C. Shao, "An Automatic Aerodynamic Shape Optimisation Framework Based on DAKOTA," *IOP Conf. Series:Materials Science and Engineering 408*, *Dalian, China*, *July 1-3, 2018*. pp. 012021.

[22] T. A. Johansen, "Optimization of a Low Reynold`s Number 2-D Inflatable Airfoil Section" Utah State University, Master Thesis, 2011.

To Cite This Article: Ş. Yiğit, S. Abuhanieh, B. Biçer, *An open-source aerodynamic shape optimization application for an unmanned aerial vehicle (UAV) propeller*, Journal of Aeronautics and Space Technologies **15**(2), 1-12 (2022).

VITAE

Şahin Yiğit received his B.Sc., M.Sc. degrees in Department of Mechanical Engineering from Karadeniz Technical University, Turkey in 2009 and 2011, respectively. He received his PhD degree in School of Engineering from Newcastle University, United Kingdom in 2018. He worked as post-doctoral researcher in the Department of Aerospace Engineering at Bundeswehr University Munich at Germany between 2018-2020. Currently, he is working in the OpenSource CFD group as a Senior CFD Engineer at the Turkish Aerospace Company.

Saleh Abuhanieh received his B.Sc. degree in Electrical Power Engineering from Yarmouk University, Jordan in 2008. He received his M.Sc. degree in Electrical Engineering from KFUPM University, Saudi Arabia in 2012. He received his PhD degree from the Mechanical Engineering Faculty at Atılım University, Turkey in 2022. He is working at the Turkish Aerospace Company in the OpenSource CFD group as a Senior CFD Engineer.

Barış Biçer received his B.Sc. degree and first master in Faculty of Naval Architecture and Ocean Engineering from İstanbul Technical University, Turkey in 2010. He received his second M.Sc. degree and PhD in the Science & Energy Lab from engineering program of Kobe University, in Japan 2015. He is founder of Turkish OpenFOAM USER's Group and OpenSource CFD Team in Turkish Aerospace. He is working in the OpenSource CFD group as Chief Engineer at the Turkish Aerospace Company.