The best efficiency point of an axial fan at low-pressure conditions

Jose J Corona Jr1, Osama Mesalhy2, Louis Chow3, Quinn Leland4 and John P Kizito1

Abstract
In the current work, the objective is to determine the best efficiency point (BEP) of an axial fan using CFD. Analyzing the performance of the fan based upon the parameters chosen can lead to the optimal design of an axial flow fan for aerospace applications where the ambient pressure varies rapidly. The 2-bladed fan chosen for the study is the Propimax 2L which is considered the base fan used for comparison of all the results of the work. The set of parameters tested were fan rotational speed, ambient pressure conditions, blade count, and the airfoil design. All the performance measures were based on overall fan efficiency. The results yield the following: an increased rotational speed led to higher efficiencies, the most efficient ambient pressure of which the fan can perform is 0.7 atm, a 5-bladed fan configuration produced the highest efficiency, and airfoil selection is critical for fan efficiency enhancements. The results demonstrated that at 0.7 atm the fan efficiency is the highest due to the changes in power consumption to the density effect. A key finding in the work is that higher blade counts do not necessarily lead to higher performing axial fans. A high cambered airfoil provided a higher flow rate at free delivery than that of the Propimax 2L design, but the rotorcraft airfoil did not yield favorable results. The analysis is focused on the fan design of cooling of the electromechanical actuators (EMAs).

Keywords
Axial fan, computational fluid dynamics, variable ambient pressure conditions, airfoil design, best efficiency point

Introduction
In the current work, the focus is to study the effects of turbomachinery, specifically axial fans, for various parameter changes. Axial fans can deliver high volumes of air at low static pressures. The rotational speed is a key parameter used to determine the effectiveness of an axial fan. In the automotive industry turbocharger systems (compressor and turbine) are coupled and the rotational speed at which those systems are running determines the components’ peak performance. Martin et al.1 maximum efficiencies of these turbomachine systems are highly dependent on rotational speed. A study demonstrates that for turbines used in solar-powered chimney power plants the overall efficiency of the turbine reaches a maximum at a certain rotational speed.2 Not only is the aerodynamics critical for the efficiency of the fan, but also the electric motors that drive the fans. Li et al.3 mapping of electric motor for the specific application is critical since loading ratio and speed

1North Carolina Agricultural and Technical State University, Greensboro, NC, USA
2Zagazig University, Ash Sharqia Governorate, Egypt
3University of Central Florida, Orlando, Florida, USA
4Power and Control Division, Aerospace Systems Directorate, Air Force Research Laboratory, Wright-Patterson AFB, OH, USA

Corresponding author:
John P Kizito, North Carolina Agricultural and Technical State University, 1601 East Market Street, Greensboro, NC 27411, USA.
Email: jpkizito@ncat.edu

Creative Commons CC BY: This article is distributed under the terms of the Creative Commons Attribution 4.0 License (https://creativecommons.org/licenses/by/4.0/) which permits any use, reproduction and distribution of the work without further permission provided the original work is attributed as specified on the SAGE and Open Access pages (https://us.sagepub.com/en-us/nam/open-access-at-sage).
ratio are optimized and best predicted at their rated performance measures.

The density effects of the ambient drastically affect the performance of the fans. The fan performance greatly deteriorates from an atmospheric pressure condition (1 atm) to the high-altitude conditions (0.2 atm). Researchers have currently conducted studies of these harsh changes in the environment on fan performance.\textsuperscript{4–10} Corona et al.,\textsuperscript{4} Wu et al.,\textsuperscript{9} fan scaling laws were tested and demonstrated to accurately predict fan performance based upon scaling of fan speed, density, and power input except for the low pressure (0.2 atm) condition. Reliability increase based upon multiple similar fans has been conducted to ensure proper cooling to the electric motors in EMAs.\textsuperscript{8,10}

Many CFD works have been performed using traditional fan airfoil shapes. Jayaram Thumbe\textsuperscript{11} a traditional curved flat plate airfoil is implemented to determine the fan performance curve of a 6-bladed fan; moreover, the researchers optimized the original 7-bladed design to that of a 6-bladed design. Aeroacoustics is a vital parameter for the design of more efficient blades since energy is more efficiently converted without some useful energy being transferred into noise pollution by the fan.\textsuperscript{12,13}

Other works regarding CFD of axial fans have focused on obtaining the best model to simulate the effects of the flow across the fan. Gullberg et al.\textsuperscript{14,15} correction methods are introduced using the fan scaling laws to obtain the high volumetric flow rate conditions to correlate well with experimental data. The correction methods were scaled using the fan affinity laws for the results to match the experimental results. The use of the affinity laws is to determine an approximation of the pump’s performance at various conditions; moreover, the affinity laws have limitations based upon the size of the pump,\textsuperscript{16} pump efficiency,\textsuperscript{17} pump efficiency cannot be predicted well in terms of power and cannot be used for predict extreme power-saving.\textsuperscript{18} CFD analysis of fan arrays has been conducted by Yu and Chen\textsuperscript{19} to supply intake air for a wind turbine, but the researchers discovered that placing the fans far downstream would provoke an unwanted swirl that drops the efficiency of the wind turbine. Other geometrical considerations such as low hub-to-tip ratios have been accurately predicted using careful turbulence modeling, boundary conditions, trailing edge shape, and areas of non-aerodynamic shape sections at low rotational speeds.\textsuperscript{20} Nazmi Ilikan and Ayder,\textsuperscript{21} CFD is used as a tool to determine the effects of backward sweep and forward sweep angles in blade design; moreover, it was determined that forward sweep performed better at low rotational speeds but backward sweep fans' performance deteriorated.

As of recent, a more aerodynamic thought has been implemented into fan design. Using low fidelity software (i.e. X-FOIL) can provide insight with accurate results in airfoil for improved efficiency and aerodynamics for blade design.\textsuperscript{22} Other researchers have focused on creating techniques of solver based optimization strategies to design axial fans combining CFD and artificial neural network (ANN) of the multilayer perceptron (MLP) and has been designed, tested, and complimented their results for nine propeller designs using the two techniques.\textsuperscript{23} Optimization techniques based upon the gradient method have been also validated to work on axial fans to improve fan efficiencies and operating pressures in other works.\textsuperscript{24}

To the best of the authors’ knowledge, the work before the current work has not been conducted to this extent. The work presented is an extension of the work that has been conducted\textsuperscript{4–7,9,25,26} with an emphasis on analyzing the best efficiency points of the fan at various fan rotational speeds, ambient pressure conditions, blade count, and the effect of interchanging the preexisting design with an airfoil. The current work can be used as a basis for considering the best operating condition of fan efficiency based upon the parameters presented in this study. The work is tailored to cooling EMAs since there is wide support in designing more electric/all-electric aircraft as the future of aviation for numerous reasons.\textsuperscript{27} The fans analyzed in the current work are intended for cooling purposes of the electric motors found in the EMAs. The EMAs are found within the wing and have small enclosures\textsuperscript{28} which is why a 70 mm fan is chosen for the analysis.

Mathematical model

The approach taken in the present work is the Multiple Reference Frame (MRF) approach to model the interaction of the rotor (axial fan) and stator (channel) in the numerical model. The turbulent model used to solve the flow equations was the Standard $k – \varepsilon$ turbulence model. The standard $k – \varepsilon$ turbulence model is the typical model employed by the CFD solver; additionally, a Modified Wall Function is employed to characterize laminar and turbulent flow near the wall. The CFD’s modified wall function technique provides accurate velocity and temperature turbulent boundary conditions for the conservation equations.\textsuperscript{29}

The CFD consists of three separate regions, two are non-rotating and the other region is rotating which surrounds the axial fan. The flow is steady-state and local averaging is implemented to approximate the flow interaction among the non-rotating region and the rotating region. The non-rotating region implements the inertial, non-rotating Cartesian global coordinate system for mass, linear momentum, and angular momentum are shown as,\textsuperscript{29,30}

$$\frac{\partial}{\partial x_j} (\rho u_i) = 0$$  \hspace{1cm} (1)
\[
\frac{\partial}{\partial x_i} (\rho u_i u_j) + \frac{\partial p}{\partial x_i} = \frac{\partial}{\partial x_j} (\tau_{ij} + \tau_{ij}^R) + S_i = 1,2,3 \tag{2}
\]
\[
\frac{\partial}{\partial x_i} (\rho H u_i) = \frac{\partial}{\partial x_j} (u_j (\tau_{ij} + \tau_{ij}^R)) + \frac{\partial p}{\partial t} - \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + \rho e + S_i u_i \tag{3}
\]
\[
H = h + \frac{u^2}{2} + \frac{5}{3} k - \frac{\kappa^2 r^2}{2} \tag{4}
\]
where \( u \) is the fluid velocity, \( \rho \) is the fluid density, \( h \) is the thermal enthalpy, \( \tau_{ij} \) is the viscous shear stress tensor, \( \Omega \) is the angular velocity of the rotating coordinate system, \( r \) is the distance from a point to the rotation axis of the rotating reference frame, \( k \) is the turbulence dissipation, \( \epsilon \) is the turbulent dissipation, \( S_i \) is a mass-distributed external force per unit mass of the coordinate system's rotation, and the subscripts \( i,j \) denote the various components in their respective coordinate directions. The only mass-distributed force in the CFD is due to the rotating region and gravitational and any porosity effects are neglected which indicates that the equation for the mass-distributed external force yields,
\[
S_i^{\text{rotation}} = -2 \epsilon_{ijk} \Omega \rho u_k + \rho \kappa r^2 \tag{5}
\]
where \( \epsilon_{ijk} \) is the Levy-Civita function. The CFD solver implements relaxation methods to solve the equations in the entire domain due to the rotating and non-rotating physics of the problem. The fluid in the flow region is air; therefore, the equation for Newtonians fluids are implemented to solve the viscous shear stress tensor and the Reynolds-stress tensor in the following manner,
\[
\tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) \tag{6}
\]
\[
\tau_{ij}^R = \mu_R \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \right) - \frac{2}{3} \rho k \delta_{ij} \tag{7}
\]
where the \( \mu \) is the dynamic viscosity of the fluid, \( \mu_R \) is the turbulent eddy viscosity coefficient, \( \delta_{ij} \) is the Kronecker delta function. The set of equations (1)–(9) presented in this section demonstrate the CFD equations solved to obtain the results demonstrated in the results section.

**CFD model**

**Numerical model setup**

The computational model was created using SOLIDWORKS 2019. The CFD package used in the current work is SOLIDWORKS Flow Simulation. The CFD package is written by Dassault Systemes; additionally, the commercial package uses robust techniques to accurately solve the flow/thermal fields. The software is widely used in designing and building commercial jet fighters.29–31 The model consists of a circular tube extruded for a total length, \( L = 1220mm \). The diameter of the tube is \( D_{\text{tube}} = 70mm \). The 70 mm fan casing diameter was chosen due to the size of the baseline Ametek; moreover, the current work did not analyze the effects of fan diameter on the fan performance. The CFD results were obtained using the built-in CFD package used in the software. In Figure 1 the boundary conditions of the inlet (Lid 1) and outlet (Lid 2) are demonstrated along with the fan in the center of the tube in the axial direction. The boundary conditions of the model are changed depending upon the pressure conditions in the simulation. Table 1 demonstrates the boundary conditions for modeling the various ambient conditions during the takeoff/descent of an aircraft. The working fluid is air; therefore, the software has the properties of air in-built. The changes in the air due to temperature are modeled, these key hydrodynamic/thermodynamic parameters include the specific heat \( C_p \), dynamic viscosity \( \mu \), thermal conductivity \( \kappa \), specific heat ratio \( \gamma = 1.399 \), and molecular mass \( M = 0.0290 \text{ kg/mol} \). The viscosity of the fluid is greatly affected by the temperature; moreover, the CFD analysis does consider the ambient conditions as listed in Table 1. The air is incompressible, and the solver has the fluid properties imbedded to account for the changes in the thermodynamic properties in the current work.
The solid structures in the flow are also assigned real wall boundary conditions such that the software recognizes that fluid must not pass through that region of the model. The inner wall of the channel and the fan assembly (blades and hub) are given the real wall boundary condition. The fluid region is divided into two distinct regions a rotational and non-rotational fluid region given in Figures 2 and 3, respectively.

In Figure 3, the rotational region is highlighted in blue and the green arrow depicts the counterclockwise direction of the movement of the fluid thus causing the flow to travel into the axis of the page.

In the current study, various parameters were tested. Firstly, a 2-bladed fan (Ametek Propimax 2L)\textsuperscript{32} was tested for various rotational fan speeds, $\Omega = (10000, 21170, 30000, 40000)\text{rpm}$, at standard temperature and pressure conditions. Following the fan speed study, the ambient pressure was throttled as demonstrated in Table 1 for the various pressure conditions. Thirdly, blade count was studied to find the best configuration for the fan’s geometric blade parameters. The fan configurations demonstrated in Figure 4(a)–(e) have the same hub and tip diameter of $D_{\text{hub}} = 25\text{mm}$ and $D_{\text{tip}} = 68.5\text{mm}$, respectively.

Lastly, a comparison of airfoil design was considered. A high lift airfoil was chosen for the analysis against that of the traditional airfoil design of the fan. The Selig S1210 and Sikorsky Flemming SSC-A07 airfoils were downloaded from the UIUC airfoil database\textsuperscript{33,34} and substituted the spline shape of the Propimax 2L blade; moreover, Figure 5 demonstrates the three fan designs next to each other. The Propimax 2L fan CAD is used as a guide to trace along and replace the airfoil design by superimposing the S1210 and SSCA07 CAD over the Propimax 2L to ensure that the geometry is imitated as closely as possible. By superimposing the new CADs on top of the old CAD it is assured that the only parameter varied from one simulation to the next is the airfoil without any drastic changes to the other geometrical blade parameters of the impeller. In the current work, the fan’s size is small enough to model the entire structure without the
requirement of excessively large computational resources. At the expense of larger computational time, modeling the full scale 3D fan provides the best accuracy. The grid independent analysis demonstrated that the mesh used yielded acceptable results.

**Grid independence studies**

The CFD uses a Cartesian mesh to solve all problems in the software. The CFD can generate a mesh-based upon the imported CAD geometry created within SOLIDWORKS. In this work, the mesh is set at an initial mesh and allowed to refine as the calculation progresses at preset intervals within the calculation. In Table 2, the refinement intervals are specified and the cell count of the mesh for the various fan geometries. The overall global computational domain is set to medium size grid density, (automatic setting 3), two localized grid clustering techniques are utilized to best capture the areas where the grid requires higher refinement. Refinement at the wall of the tube is applied; moreover, the CFD recognizes that the wall is solid (due to real wall boundary condition) and an option of refining the grid near the wall is applied with an equidistant grid technique to cluster more computational squares in that region. The other region that has an initial grid concentration is fluid surrounding the fan with a three-stage equidistant technique. In this case, three stages were chosen to get a progression in the grid near the wall of the fan to the global grid of the entire domain. An advanced channel refinement is also applied to this region of the domain with a maximum refinement of three levels to accurately discretize the grid surrounding the fan. Moreover, a solution-adaptive grid is applied to the computational domain, the various refinement steps allow the grid to change from a lowly populated grid size to a more robust grid to better capture the areas of high field gradients within the domain. Therefore, based upon this grid refinement strategy, the CFD populates the domain as shown in Table 2.

The travel is defined as the number of iterations required for the propagation of a disturbance across the entire computational domain. To save computational time without compromising the results, a total travel time of four is assigned with the refinement steps of two and three to obtain the results that match closely with the manufacturer’s specified curve given in the following section of the paper. In Figures 6 and 7, the final computational domain is shown for the 2-bladed fan in XY-plane and YZ-plane, respectively.

The red regions demonstrate the maximum amount of refinement. The minimum refinement is navy blue which is the region away from the wall and fan. The CFD determines high field gradients surrounding the

| Refinement step (travel) | 2-bladed | 5-bladed | 7-bladed | 10-bladed | 12-bladed | 2-bladed S1210 | 2-bladed SSCA07 |
|-------------------------|---------|---------|---------|----------|----------|----------------|----------------|
| Initial                 | 96,0351 | 9,80,330 | 9,66,966 | 10,17,866| 10,53,136 | 3,71,957       | 5,37,570       |
| 2                      | 10,69,436 | 11,03,150 | 10,91,956 | 11,38,629 | 11,65,820 | 8,20,596       | 6,28,254       |
| 3                      | 21,51,746 | 22,49,585 | 22,54,195 | 23,02,801 | 24,56,555 | 22,16,378      | 20,78,414      |

**Figure 5.** Side view of the three airfoil shapes.

**Figure 6.** A cut plot demonstrating the grid in the XY-plane intersecting the fan.
Validation of model

The Propimax 2L fan curve provided by the manufacturer curve was compared to the static pressure difference across the fan-generated by the CFD model described in the previous section. In Figure 8, the validation of the numerical model was determined. The numerical model described in the previous section demonstrated to agree with the manufacturer’s curve for the 2-bladed fans. All the pressure points lie within 2% of the manufacturer’s data except for 500 Pa. The validation curve demonstrated good agreement with the manufacturer’s specification curve at low flow rates with exception to 0.5 m³/min. Regardless, fitting the data with quadratic demonstrated that the two curves were positioned very close to each other. The dotted lines demonstrate a quadratic fit of the manufacturer’s specification data. Similarly, the dashed line is a quadratic fit of the numerical data obtained from the CFD results. The validation of the curve was performed at 21,170 rpm; moreover, it is the fan curve provided by the manufacturer for the 2-bladed fan. The numerical model follows the trend of the data provided in the literature.\(^{32}\)

Results and discussion

The results presented in this work will be in nondimensional coefficients of capacity, head, power, and fan efficiency. Presenting the results in nondimensional terms provides a measure of comparison among the parameters to obtain a better representation of the fan’s performance. The equations were carefully derived from Buckingham Pi theory; therefore, the following equations denote the coefficients of head, capacity, power, and efficiency, respectively\(^ {36,37} \)

\[
C_H = \frac{10FSP}{\rho \Omega^2 D^2} \\
C_Q = \frac{10m}{\rho \Omega^2 D^3} \\
C_P = \frac{100T_m}{\rho \Omega^2 D^5} \\
\eta = \frac{C_Q C_H}{C_P}
\]

where \(FSP = \rho g H_p\) and fan static pressure, gravitational constant \(g = 9.80 \text{ m/s}^2\), the \(m\) is the mass flow rate of which both results were obtained from the CFD results from the entrance and exit of the computational domain, \(T_m\) is the magnitude of the torque of the entire fan including impeller and hub from the CFD. The maximum Mach number calculated was \(Ma = 0.52\), for the highest rotational speed and the lowest ambient condition. These equations are applicable to compressible flows to the use of Buckingham Pi theorem to the design, analysis and characterization of propeller in addition to blowers.\(^ {37,38} \)

Results at variable rotational speed

In Figure 9, the efficiency curve is plotted as a function of the capacity coefficient. The data demonstrated for each of the fan performance curves have 5–6 data points; moreover, using those points the efficiency curves were determined with quadratic fits that demonstrated high coefficient of determinations, \(R^2\geq 0.98\). From Figure 9, the effects of higher rotational fan speeds indicate to enhance the BEP; thus, indicating that the fluid power produced is greater than the power required to operate the fan at the higher speeds. The capacity coefficient is multiplied by a factor of 100 for plotting purposes for the reader to interpret the data. The arrows indicate the best efficiency points (BEP) of
the 2-bladed fan for the maximum performance fan speed, $\Omega = 40,000rpm$, and the minimum performing rotational speed, $\Omega = 10,000rpm$. The BEP shifts only slightly for such a wide range of rotational speeds. The static backpressure assigned at the downstream lid was throttled and by trial and error until an acceptable calculation was determined. The zero-efficiency point was determined since the back pressure assigned calculated an extremely small volumetric flow rate within the system. The 2-bladed fan’s operating point occurs at approximately the same volumetric flow rate, but with peak efficiencies that vary from $\eta = 0.155 - 0.185$. In terms of efficiency, the fan results indicate that the performance as a function of the rotational speed of the fan does not vary drastically. Operating the fan at a much higher fan speed will not lead to a more efficient fan. In Table 3, the maximum efficiency is taken for the results presented in Figure 8. The results demonstrate that decreasing the fan speed to 10,000 rpm demonstrate a 6% drop in BEP. Increasing the rotational speed past the baseline increases the BEP to 10% and 12% for 30,000 rpm and 40,000 rpm, respectively. Demonstrating that increasing the rotational speed by an additional 10,000 rpm does not yield a considerable performance upgrade in terms of efficiency. An increase from 21,170 to 30,000 rpm results in an increase of 10%. However, a change from 30,000 to 40,000 rpm yields an increase of approximately 3% in efficiency. From 21,170 to 40,000 rpm results in an increase of 13%. The tip clearances numerically tested could affect the efficiencies of the axial fans. In the current work, the tip clearance was not a parameter that was not analyzed. The tip clearance was not altered since the numerical results correlated well with the manufacturer’s data in the validation curve, in Figure 8.

In Figure 10, the BEP was marked by the black vertical. The rotational fan speed for Figure 10 was for 40,000 rpm and was chosen due to that fan speed having the highest efficiency demonstrated in Figure 9. The diamonds indicate the head coefficient, triangles are the power coefficient, and the squares demonstrate the efficiency, and these three parameters are all function of the capacity coefficient. The best efficient power coefficient, head coefficient, and capacity coefficient for 40,000 rpm is $100C_p^* = 0.18$, $100C_H^* = 0.11$, and $100C_Q^* = 3.48$, respectively. Past the BEP, the coefficient of power of the fan saturates, while the head coefficient continues to decrease as the capacity coefficient increases. The throttling of the backpressure increases/decreases the resistance of the system; thus, the additional resistance affects the fan’s capacity to move the fluid through the passage. The coefficient of power remains constant since the rotational speed of the fan and the torque determined remain constant past the BEP. The fan is designed for 21,170 rpm at 1 atm. The purpose of varying the speed up to 40,000 rpm, is to determine performance at low pressures. The numerical prediction indicates that an increase in rotational speed of the fan increases the BEP. Whereas the manufacturer’s suggested speed is 21,170 rpm (1 atm); we searched for conditions needed for our test plan. The current set of data indicates that an increase in rotational speed has the potential to improve the BEP of the fan at higher rotational speeds. The use of finite element analysis (FEA) can provide predictions on the stresses and elongations of the blades at 40,000 rpm; regardless, the current work focused on analyzing the

Table 3. Efficiency curves at various rotational speeds of the fan.

| Fan speed [rpm] | $\eta$ | % change |
|----------------|-------|----------|
| 10,000         | 0.155 | -6.02    |
| 21,170         | 0.165 | n/a      |
| 30,000         | 0.181 | 9.70     |
| 40,000         | 0.185 | 12.23    |

Figure 9. Efficiency curve of the 2-bladed fan at various rotational speeds.

Figure 10. BEP of the fan at 40,000 rpm.
Results at variable ambient pressure conditions

The effects of pressure are analyzed in this work to determine the reaction of the fan’s performance to the changes in the ambient pressure. The density of the fluid is directly affected by the changes in the ambient pressure; moreover, as aircraft ascends to cruising altitudes the pressure is directly affected by the elevation change. The following relationships are used to determine the temperature and pressure changes due to the elevation,

\[ T = ay + T_s \]  
\[ P = P_s \left( \frac{T}{T_s} \right)^{-B/RT} \]

where \( a = -6.5 \frac{K}{\text{km}} \) is the lapse rate, \( y \) is the position in altitude, \( T \) is the temperature at \( y \), \( T_s = 288.16K \) the temperature at sea level, \( R = 287 \frac{K}{\text{mbar}} \) is the gas constant of air, \( P_s = 101.325\text{kPa} \) the pressure at sea level. Equations (12) and (13) provide the pressure and temperature at any given point in the troposphere with less than 0.1% loss in accuracy of the measured experimental values found in the literature.

In Figure 11, the efficiency curves for variable ambient pressure conditions is demonstrated. There is a broad range of BEP for the fan running at 21,170 rpm when the pressure within the system changes from atmospheric and 10% of atmospheric pressure. The lower density at 0.7 atm yields increased fan performance; moreover, the ambient has a lower resistance due to a lower amount of fluid present, but with a sufficient amount that still benefits the fan’s airflow delivery. At 70% atmospheric pressure (hollow diamond) the highest efficiency is determined. After 0.7 atm, the pressures with decreasing efficiency are 0.5 atm (squares), 1 atm (solid diamond), 0.2 atm (x), and 0.1 atm (cross). The highest efficiency determined was \( \eta = 0.224 \) and \( \eta > 0.113 \) for 0.7 atm and 0.1 atm, respectively. The results establish that there is about a 10% difference between the highest and lowest efficiency for a broad range of ambient pressures. The capacity coefficient from the highest to lowest BEP demonstrates a spread of \( 100C_Q^{0.224} \approx 2.9 - 3.8 \). In Table 4, the BEP at various ambient pressure conditions is presented. The baseline pressure is the atmospheric pressure condition and all the remaining pressures are compared to 1 atm. The 2-bladed fan operates the best at 0.7 atm and demonstrates a 37% increase in the BEP over the atmospheric condition. At 0.5 atm, the fan performance increases by 9% at the BEP. Additionally, the 0.2 atm (40,000 feet) and 0.1 atm (50,000 feet) pressure conditions have drops percent changes in maximum efficiency by 2% and 31% compared to the 1 atm condition, respectively. At extremely low pressure conditions of 0.1 atm, the BEP of the fan drops drastically compared to the standard atmospheric pressure conditions; therefore, using the current fan configuration at 0.1 atm demonstrates undesirable operating conditions. Most civilian aircraft have the ceiling of 40,000 feet. As the pressure in the system changes from 1 atm to 0.1 atm, the first BEP demonstrates an increase and then have a severe drop in BEP at the extremely low-pressure conditions of 0.1 atm.

In Figure 12, the ambient pressure at which the fan has the highest efficiency is 0.7 atm. The density effects at various ambient pressures causes the highest efficiency to occur at 70% of atmospheric. The density and mass flow rate are directly related, and a decrease in density affects the mass flow rate that the fan can deliver. Conversely, the less dense air provides less resistance to the fan which means the air can be easily circulated at the 70% atmospheric condition. At 0.1 atm the low pressure and temperature conditions have adverse effects to the BEP of the fan. The optimal coefficient of power and the head coefficient for 0.7 atm

| Ambient pressure [atm] | \( \eta \) | % change |
|------------------------|--------|---------|
| 1                      | 0.164  | n/a     |
| 0.7                    | 0.224  | 36.79   |
| 0.5                    | 0.179  | 9.15    |
| 0.2                    | 0.160  | -2.40   |
| 0.1                    | 0.113  | -30.96  |

Table 4. BEP determined at variable ambient pressure conditions.
and 21,170 rpm is $100C_p^* \approx 0.18$ and $10C_H^* \approx 0.11$, respectively. The corresponding capacity coefficient to the BEP is around $100C_Q^* \approx 3.8$. Typical propeller driven aircraft cruise at 10,000 ft (~0.7 atm) which exhibit the lowest compressibility losses which in turn will produce the highest efficiencies.\(^{40}\)

### Results for variable blade count

In Figure 13, the efficiency curves for the fan at multiple blade configurations were determined. The 5-bladed fan (circles) established the highest efficiency amongst the other blade counts. The average space-chord ratio from hub to the tip of the 5-bladed is approximately 1 which indicates the optimal blade design without experiencing a blockage in the flow. All the data was fitted using a quadratic regression line and every fan count configuration demonstrated acceptable agreement to the regression line except for the 5-bladed fan. The quadratic fit does not follow the 5-bladed fan results due to the BEP’s significant increase in at the BEP compared to the other results obtained from the CFD. That peak in efficiency is reported since it reflects the performance increase of the 5-bladed fan configuration to the other cases studied. The 2-bladed fan (star), the Propimax 2L fan design, demonstrates the second-best efficient configuration followed by 7-bladed (triangles), 10-bladed (hollow diamonds), and lastly the 12-bladed fan design (crosses). The peak efficiency for the Propimax 2L blade design shown in Figure 4(b) is the 5-bladed fan design; moreover, past 5 blades the efficiency of the fan starts to deteriorate. Even a 7-bladed fan demonstrates a less efficient configuration than that of the 2-bladed fan. The efficiency drop is intensified further when the blade count is a 12-bladed fan. The BEP of the different blade counts changes drastically from the worst to the best case, $\eta = 0.05 - 0.22$. The capacity coefficient also reflects a broad range from the two extreme of $100C_Q^* = 1.7 - 3.5$ which represents nearly a capacity difference of $100C_Q^* = 2$; furthermore, determining that almost twice as much airflow delivery at the BEP for a 5-bladed fan instead of the 12-bladed fan. The BEP of the Propimax 2L fan configuration (BEP highlighted in a circle) demonstrates an efficiency and capacity coefficient of $\eta = 0.165$ and $100C_Q^* = 3.2$. In Table 5, the percent change in efficiency is determined. The 2-bladed fan is taken as the baseline and is compared to that of the other various blade counts. There is a considerable change in improvement by 30% in using the 5-bladed fan configuration. Regardless, that is the only fan count that demonstrates beneficial results to that of the 2-bladed fan. Blade counts of 10 and 12 demonstrate the adverse effects of a 58% and nearly 70% drop in efficiency, respectively.

![Figure 12. BEP with the optimal head, power, and capacity coefficients for 0.7 atm at 21,170 rpm.](image)

![Figure 13. Efficiency curves of the fan for variable blade count at 21,170 rpm and 1 atm.](image)

### Table 5. Percentage change in efficiency as the blade count is modified.

| Blade count | $\eta$  | % change |
|-------------|---------|----------|
| 2           | 0.165   | n/a      |
| 5           | 0.215   | 30.30    |
| 7           | 0.132   | -19.72   |
| 10          | 0.069   | -58.18   |
| 12          | 0.050   | -69.70   |

In Figure 14, the BEP is identified to demonstrate the most efficient point of the 5-bladed fan in terms of power, head, and capacity coefficient. The results indicated that the optimal coefficient of power, head, and capacity was, $100C_p^* = 0.34$, $10C_H^* = 0.2$, and $100C_Q^* = 0.36$. These points indicate the optimal operation of the 5-bladed fan at a rotational speed of 21,170 rpm and 1 atm.
In Figure 15, the various fan performance curves are demonstrated as the blade count is altered. The 4-point star denotes the BEP for the different fan blade configurations. The BEPs are derived from Figure 13. In terms of coefficient of pressure as a function of the coefficient of capacity, the best to worst performance is as follows, 5-bladed fan (circles), 7-bladed fan (triangles), 2-bladed fan (stars), 10-bladed fan (hollow diamonds), 12-bladed fan (crosses). Figure 15 demonstrates that the 7-bladed fan has a higher coefficient of head at the BEP and at a lower capacity coefficient than that of the 2-bladed fan. Although the 2-bladed fan demonstrates a lower operating point in terms of head capacity, the results from Figure 13 indicate that the 2-bladed fan is still more efficient at the peak operating condition. The efficiency of the fan provides a quantifiable measure for the effectiveness of the fan. The results of Figure 13 are the main parameter to consider in the analysis to determine the best blade count.

The space-chord ratio at the hub and tip for the various blade configuration is in Figure 16. In reference \(^4\) the spacing of the fan at the hub and tip is defined as,

\[ s = \frac{2\pi r}{n_b} \]  

(14)

where \( r \) is the radius at the hub/tip and \( n_b \) is the number of blades. In the current work, the geometry of the blade was unaltered except for the blade count. By changing the number of blades, the fan’s solidity was affected. The following geometric quantities were kept constant for all of the blade counts: chord length \( c = 29.2 \text{mm} \), inlet angle \( \beta_{\text{in}} = 8.2^\circ \), and outlet angle \( \beta_{\text{out}} = 32.6^\circ \). The optimal performance of the fan was numerically determined at 5 blades which is an average space-chord ratio of approximately 1. Inverse design techniques have demonstrated optimal space-chord ratios at 1.183. \(^42\) The 10 and 12 bladed fans experienced high blockage due to the small space-chord ratios at those blade counts. Lin and Chou,\(^43\) axial fan cooling performance decreased significantly with increased blockage of airflow.

In Figure 17, the BEP is plotted as a function of the estimated weight of each fan blade count. The mass of each fan can be determined using a feature provided in the software; therefore, once approximating the mass the weight of the fan can be estimated. The blade count is denoted above the data point. For aerospace applications, there is a weight penalty to consider. Based upon the BEP, it is best to use either a 2-bladed or 5-bladed configuration, and anything greater than the 5-bladed fan count would not yield beneficial results. There is nearly a 3% increase in estimated weight from the 2-bladed to the 5-bladed, but almost a 57% increase in efficiency. The results indicate that a small weight penalty is worth the increase in maximum efficiency of the fan.
Results for changed airfoil design

In the current work, a change in airfoil design was analyzed. The three airfoil designs were compared to 1 atm and 21,170 rpm. The condition chosen for further testing was selected at 21,170 rpm and standard atmospheric conditions as a result of a point of comparison to that of the original fan’s known operating conditions by the manufacturer. The Propimax 2L fan design is based upon a curved flat plate to force the fluid to travel axially across the channel. A high cambered, high lift airfoil was used to study the effects of the airfoil on the airflow to enhance the efficiency of the fan. The Selig S1210 airfoil was analyzed as shown in Figure 5 (fan on the right). The third airfoil design was based upon a typical rotorcraft blade used for helicopter blades; moreover, from Figure 5 (fan in the middle) the airfoil is the SSCA07. The hydrodynamic results for the three airfoils are demonstrated in Figure 18. In Figure 19, the efficiency curves for the three airfoils are determined and the Propimax 2L fan still performs slightly more efficiently than that of the S1210 airfoil replacement. The difference in the BEP is approximately 2% higher for the Propimax 2L fan. From Figure 19, it is demonstrated that the BEP shifts from $100C_\theta = 3.2$ for the Propimax 2L fan, to $100C_\theta = 3.7$ for the S1210 fan. The increase in airflow at a higher capacity is reflected in Figure 18; furthermore, at the BEP the best hydrodynamic performance of the three fans changes, and is determined that the S1210 airfoil demonstrates a higher performance than that of the Propimax 2L airfoil. The CFD results demonstrated that in terms of hydrodynamic and efficiency performance the rotor blade airfoil (SSCA07) performed poorly as shown in Figures 18 and 19. The lack of performance and fan efficiency for the SSCA07 airfoil is due to its lack of curvature, the results indicate that a curve in the Propimax 2L and high cambered (S1210) nature of the high lift airfoil allows the fan’s rotational behavior to force the air downstream. In the current work, the airfoils were swapped directly from the originally shaped fan; moreover, the SSCA07 airfoil’s pitch angles were not optimized. Each airfoil has an optimal angle of attack and in the current work a straight swap of blade shape does not guarantee the best performance in the other airfoils. The SSCA07 is a symmetrical airfoil compared to a high camber S1210 airfoil.

Once the Propimax 2L fan design reaches the BEP it begins to drastically drop in performance; on the other hand, the S1210 airfoil design demonstrates a better hydrodynamic performance past the Propimax 2L design. The curved plate airfoil design demonstrates a higher BEP, but when operating at low resistance conditions exhibits less performance than that of the S1210.

In Figure 20, a contour plot of the relative pressure along the axial direction of the Propimax 2L curved plate airfoil design is demonstrated with velocity vectors to demonstrate the direction of the fluid. The operating ambient conditions for Figure 20 are 1 atm and
21,170 rpm. The CFD determines the relative pressure of the overall domain. The solver determines the relative pressure plots concerning the absolute ambient pressure of the solver. A positive relative pressure is a pressure region higher than the absolute atmospheric pressure; on other hand, a negative relative pressure denotes a region of pressure less than the absolute atmospheric pressure. Relative pressure is the difference between the local pressure and the ambient pressure, \( P_{\text{rel}} = P_{\text{local}} - P_{\text{amb}} \). The scale for the results of the following figures is given the following range manually, \( P_{\text{rel}} = (-1620 - 700) \text{Pa} \). This range was chosen to best compare the flow resistances and airfoil designs.

In Figure 20, the flow comes into the suction surface of the fan linearly, but close to the wall the fluid experiences rotation due to the rotational effects of the fan. As soon as the fluid enters the rotational region the fluid rotates and continues to have that rotational effect far downstream of the fan. The relative pressure also has a slight increase across the fan from the green region to the yellow region. In Figure 21(a)–(f), the region near the fan blade is analyzed. Along with the suction pressure, there is a drop in the relative pressure that is distributed along the curved surface of the plate. The leading edge of the Propimax 2L fan contains a localized relative low-pressure zone near that section of the airfoil with a high-pressure area building upon the top surface. Both sides of 20a, demonstrate a spread of low and high pressure along the suction and pressure surfaces, respectively. In Figure 21(b), the high cambered airfoil design demonstrates pressure areas of relatively high and low pressure. The S1210 airfoil demonstrates a large blue region (low relative pressure) along the suction side that is halfway down the top chord line of the airfoil. For the SSCA airfoil Figure 21(c), the low-pressure zone is seen closer to the leading edge with less low-pressure regions on the suction side of the airfoil as on the S1210 airfoil. An accumulated relatively high-pressure spot at the leading edge of the S1210 airfoil can force the fluid to travel from that region of high relative pressure to the low relative pressure region down the top surface of the airfoil and along the axial direction; on the other hand, the SSCA07 airfoil demonstrates a high pressure on the leading edge of the airfoil which can lead to an area of flow resistance that affects to fan’s ability to direct the flow axially down the tube. Additionally, the pressure side of the S1210 airfoil displayed an area of high pressure in the cambered region of the airfoil. The accumulation of high pressure along the cambered area is due to the physics of the motion of the fluid as the surface is pushing the fluid in the direction of the rotation. Figure 21(a)–(c) are for the free delivery condition, in an area of the fan curve where the fan is experiencing zero system resistance and the fluid can flow without any opposition. In Figure 21(a) the tangential (upward) direction indicating the rotation of the fan and the axial direction (rightward) specifying the overall direction of the fluid flow.

In Figure 21(d)–(f) the relative pressure contours are demonstrated at high system resistance conditions. For the curved flat plate airfoil, an area of relative low-pressure recirculating flow occurs at the leading edge of the airfoil. The pocket of recirculation can lead to a drop in the efficiency of the fan. At high flow resistance, the area of high-pressure buildup along the pressure side of the airfoil is large and engulfs the region downstream of the fan shown in Figure 21(d). The S1210 airfoil demonstrates a large region of relatively low pressure at the suction side that essentially covers the entire top surface of the blade. Due to the nature of the rotational flow and the relatively high pressure on the suction surface of the blade, the fluid can experience flow reversals at the leading edge of the airfoil. The vectors indicate that the fluid driven by the impeller can recirculate back, combined with that pocket of relatively low pressure can lead to adverse effects in the air movement along the axial direction. In the bottom right corner of Figure 21(e), the flow encounters a recirculating flow. This region of recirculation causes the flow to reverse; therefore, is a cause for the fan to have to do more work to force the fluid downstream. This recirculation bubble can result from the natural phenomenon of airfoil theory where trailing edge vortices are prone to occur. Figure 21(f) displays the greatest amount of flow vortices in the streamlines of the contour plot; furthermore, the pockets of recirculation lead to the SSCA07 airfoil performing inferiorly to the other two airfoil designs. For all the airfoil shapes, the high system resistance does not allow the fluid to move freely along the axial direction; therefore, leading to the velocity vectors pointing upwards in Figure 21(d)–(f) while the vectors in the free delivery, Figure 21(a)–(c) all indicate that the flow can travel easily downstream. Previous work has been conducted to show that cambered airfoils require less power to rotate the blades for
vertical axis wind turbines; furthermore, the current work demonstrated the potential benefit of using high-cambered airfoils for axial fans.

The flow is analyzed for the curved flat plate airfoil, S1210, and SSCA07 airfoil in Figures 22 and 23, respectively. In all figures, the fans are demonstrated at atmospheric conditions, 21,170 rpm, and free delivery. The vertical black lines upstream and downstream of the fan denote the one diameter distances from the fans to give a perspective of the swirl imparted by the various fan airfoil designs. The color bar is adjusted automatically to the maximum and minimum values within the domain. The trajectories are demonstrated to flow from the inlet of the domain with 50 fluid particles to travel down the channel. In Figure 22, the curved flat plate airfoil geometry causes a clustering of the fluid particles at the suction side of the fan. The fluid velocity, $V = (0.96 - 41.06) \frac{m}{s}$ demonstrates to have regions of nearly zero velocity and high-velocity recirculation at the intake. A significant loss of energy

---

**Figure 21.** (a) Propimax 2L airfoil relative pressure contours when operating at free delivery, (b) S1210 airfoil relative pressure contours for the free delivery of the fan, (c) Relative pressure contours of the SSCA07 at free delivery at standard ambient conditions, (d) Relative pressure contours of the Propimax 2L airfoil with a back pressure of 450 Pa, (e) Relative pressure contours of S1210 airfoil with a back pressure of 450 Pa, and (f) Relative pressure contours of the SSCA07 airfoil with a back pressure of 450 Pa.
is evident due to the fluid velocity decreasing from those high-velocity zones \( V = 41.06 \text{ m/s} \) to a downstream velocity of \( V = 10.68 \text{ m/s} \) which is a 74% drop in fluid velocity.

In Figure 23, a similar analysis is conducted for the S1210 airfoil operating at the same conditions such as in Figure 22. The S1210 airfoil with its cambered design can draw the fluid evenly and imparts its rotational energy more cleanly than the curved plate airfoil. The S1210 airfoil still has a reduced recirculating flow pattern demonstrated in the maximum velocity regions of the flow. The fluid velocities in the region range from, \( V = (8.98 - 43.62) \text{ m/s} \) which demonstrates that the fan can impart energy to the air better than the curved plate geometry. The difference in maximum kinetic energy loss of the fluid energy is 69% in velocity drop.

As opposed to the S1210 airfoil, the SSCA07 airfoil demonstrates inferior results. Figure 24 exhibits the flow trajectories of the SSCA07 airfoil at free delivery. The flow field demonstrates that the air comes in uniformly into the intake side of the fan but indicates regions of flow recirculation that lead to the low capacity in volumetric flow rates compared to that of the other airfoil designs. The geometry of the SSCA07 airfoil causes the air to deflect close to the leading edge of the airfoil; moreover, the deflection of the fluid particles causes them to reverse producing localized high-velocity spikes and such a broad range in fluid velocities of the air, \( V = (4.37 - 66.56) \text{ m/s} \). Using the same metrics in comparing downstream velocity to that of the peak velocity determined in the fluid particles, the SSCA airfoil indicates to have a 99% loss in fluid velocity. The data indicates that the SSCA07 is not an ideal fan design due to its lack of curvature of the blade.

The CFD indicates that an S1210 airfoil is a viable option if the fan is going to be operated at high volumetric flow rates, (i.e. near its maximum capacity) such that it can outperform the Propimax 2L curved flat plate design. In Figure 25, the fan performance of the S1210 and original airfoil are compared at the ideal operating conditions, 40,000 rpm, and 0.7 atm. As stated previously from the flow trajectories in Figure 23, the airflow is more streamlined for the S1210 airfoil than the flow of the original airfoil; moreover, the use of a high-cambered airfoil at the free delivery conditions yields beneficial results.

From Figure 25, the performance for the high-cambered airfoil dominates the original airfoil performance at high rotational speeds of 40,000 rpm. The CFD results indicate that the S1210 airfoil’s high-cambered geometry yield a significant increase in hydrodynamic performance due to its ability to push
the air more freely at the higher rotational speeds; on the other hand, the original airfoil does not exhibit an aerodynamic benefit due to its geometry.

In this work, the main objective is to provide as much forced convection on the motor of the EMA for short bursts of time. The EMAs in aircraft will only generate high heating loads for short intervals of time.45 Moreover, the S1210 airfoil is a practical design since the fan can deliver high volumetric if the space in which the fan is operating has little obstructions impeding its delivery of airflow.

Conclusion

For the work in the paper, the 2-bladed fan by Ametek Rotron was used as the base fan for many different conditions, and geometrical alterations were studied. CFD is a tool to determine the best performing fan based on the BEP. The model was validated by comparing the results to the manufacturer’s curve provided. We have analyzed the axial Reynolds numbers. The Reynolds numbers were determined to range from laminar to turbulent flow conditions (57.6 < Re < 72452.4) depending upon the fan’s rotational speed, ambient conditions, system resistance, etc. All results were obtained using the validated model to obtain the BEP-parameterizing the fan rotational speed, the ambient pressure conditions, the fan blade count, and the variation of the airfoil. The results were carefully analyzed, and the following conclusions were reached:

1. The higher fan rotational speed provided superior efficiency results (i.e. 40,000 rpm).
2. At 0.7 atm the fan demonstrates to perform at its peak BEP greater than all the other ambient pressures; moreover, the 0.1 atm pressure condition demonstrates poor performance,
3. The best blade count for the base geometrical conditions is the 5-bladed fan design over all the others considered in this work,
4. The S1210 high-cambered airfoil did not yield a higher BEP at lower rotational speeds (21,170 rpm) but at higher rotational speeds (40,000 rpm) the increase in aerodynamic performance is drastically enhanced due to the geometry of the airfoil,
5. The SSCA07 rotorcraft airfoil was outperformed by the previous designs due to its lack of curvature to accelerate the fluid effectively to force it downstream.

For future work, we are interested in analyzing more airfoil designs to determine the best airfoil design that can outperform the current design fan in all conditions not only at the high volumetric flow rate conditions. The results presented in this work lead us to conclude that choosing a high cambered airfoil with modifications to the leading edge will yield beneficial results. Optimizing an airfoil design to maximize the efficiency of an axial flow fan will reduce the input power driving the fan, and increase the fluid power; therefore, increasing the capability of cooling EMAs for aerospace applications. Designing a fan that can efficiently cool an EMA at variable pressure is of high importance due to the current push to design all-electric aircraft/more electric aircraft.

Declaration of conflicting interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: “This material is based on research sponsored by the United States Air Force under agreement number FA8650-17-2-2230. The U.S. Government is authorized to reproduce and distribute reprints for Governmental purposes notwithstanding any copyright notation thereon.” “The views and conclusions contained herein are those of the authors and should not be interpreted as FA8650-17-2-2230 necessarily representing the official policies or endorsements, either expressed or implied, of the United States Air Force or the U.S. Government.”

ORCID iD

Jose J Corona Jr https://orcid.org/0000-0001-5855-8734

References

1. Martin G, Talon Y, Higelin P, et al. Implementing turbo-machinery physics into data map-based turbocharger models. *SAE Int J Engines* 2009; 2: 211–229.
2. Guo P, Li J, Wang Y, et al. Numerical study on the performance of a solar chimney power plant. *Energy Convers Manag* 2015; 105: 197–205.
3. Li Y, Liu M, Lau J, et al. A novel method to determine the motor efficiency under variable speed operations and partial load conditions. *Appl Energy* 2015; 144: 234–240.
4. Corona JJJ, Kwarteng AA, Warwick G, et al. *Testing of a cooling fan for wing-bay electro-mechanical actuators*. Paper no. V002T014A016. New York: ASME, 2018.
5. Mesalhy O, Rath C, Rini D, et al. A parametric fin structure design study for cooling aerospace electro-mechanical actuators with high-speed axial fans. *Heat Mass Transf* 2019; 56: 1565–1577.
6. Wu W, Lin YR, Chow LC, et al. Electromechanical actuator fan failure analysis and safety-critical design. *IEEE Aerosp Electron Syst Mag* 2017; 32: 18–25.

7. Wu W, Lin Y-R, Chow L, et al. Electromechanical actuator cooling fan blades design and optimization. SAE technical paper 2016-01-1994, 2016.

8. Wu W, Lin Y-R, Chow L, et al. Electromechanical actuator cooling fan reliability analysis and safety improvement. SAE technical paper 2016-01-1997, 2016.

9. Wu W, Lin Y-R, Chow LC, et al. Fan performance characteristics at various rotational speeds and ambient pressures. SAE technical paper 2014-01-2219, 2014.

10. Mesalhy O, Elsayed ML, Corona JJ Jr, et al. Study of a high-reliability dual-fan system for cooling aerospace electromechanical actuators. *Therm Sci Eng Prog* 2020; 18: 100540.

11. Thumbe J and Jyothish V. Analysis of six bladed axial fan using ANSYS. *Int J Eng Res Manag Stud* 2017: 68–75.

12. Schnell R. Investigation of the tonal acoustic field of a transonic fanstage by time-domain CFD-calculations with arbitrary blade counts. In: *ASME turbo expo 2004: power for land, sea, and air*, Vienna, 14–17 June 2004, pp.1763–1773. New York: ASME.

13. Kröger G, Schnell R and Humphreys ND. Optimised aerodynamic design of an OGV with reduced blade count for low noise aircraft engines. In: *ASME turbo expo 2012: turbine technical conference and exposition*, Copenhagen, 11–15 June 2012, pp.407–418. New York: ASME.

14. Gullberg P, Löfdahl L, Adelman S, et al. A correction method for stationary fan CFD MRF models. SAE international 2009-01-0178, 2009.

15. Gullberg P, Löfdahl L, Adelman S, et al. An investigation and correction method for stationary fan CFD MRF simulations. SAE international 2009-01-3067, 2009.

16. Simpson AR and Marchi A. Evaluating the approximation of the affinity laws and improving the efficiency estimate for variable speed pumps. *J Hydraul Eng* 2013; 139: 1314–1317.

17. Walski T, Zimmerman K, Dudinyak M, et al. Some surprises in estimating the efficiency of variable-speed pumps with the pump affinity laws. In: *World water & environmental resources congress 2003*, Philadelphia, PA, 23–26 June 2003, pp.1–10. New York: ASCE.

18. Ford RW. Affinity laws. *ASHRAE J* 2011; 53: 42–43.

19. Yu L and Chen Y. CFD analysis for a set of axial fan array to produce inflow for wind turbine model test. In: *The 29th international ocean and polar engineering conference*, Honolulu, HI, 16–21 June 2019, p.9. New York: International Society of Offshore and Polar Engineers.

20. Wang J and Kruyt N. Validation of CFD simulations of aerodynamic performance of low speed axial fans with low hub-to-tip ratio. In: *AlAA Scitech 2020 forum*, Orlando, FL, 6–10 January 2020. New York: American Institute of Aeronautics and Astronautics.

21. Nazmi Ilikan A and Ayder E. Influence of the sweep stacking on the performance of an axial fan. *J Turbomach* 2015; 137: 061004.

22. Zhang S, Li H, Jia W, et al. Multi-objective optimization design for airfoils with high lift-to-drag ratio based on geometric feature control. *IOP Conf Ser Earth Environ Sci* 2019; 227: 032014.

23. Bamberger K and Carolus T. Development, application, and validation of a quick optimization method for the class of axial fans. *J Turbomach* 2017; 139: 111001.

24. Cho C-H, Cho S-Y and Kim C. Development of an axial-type fan with an optimization method. *Front Energy Eng China* 2009; 3: 414.

25. Wu W, Lin Y-R and Chow L. A heat pipe assisted air-cooled rotary Wankel engine for improved durability, power and efficiency. SAE technical paper 2014-01-2160, 2014.

26. Wu W, Lin YR, Gyasi E, et al. Design and experiment on aircraft electromechanical actuator fan at different altitudes and rotational speeds. SAE international 01-12-01-0003, 2019.

27. Qiao G, Liu G, Shi ZH, et al. A review of electromechanical actuators for more/all electric aircraft systems. *Proc IMechE C J Mech Eng Sci* 2018; 232: 4128–4151.

28. McCarthy K, Heltzel A, Walters E, et al. A reduced-order enclosure radiation modeling technique for aircraft actuators. SAE international 2010-01-1741, 2010.

29. Dassault S. Technical reference solidworks flow simulation. 2019.

30. Sobachkin A and Dumnov G. Numerical basis of CAD-embedded CFD. 2014. Solidworks.

31. Dassault S. Technical reference solidworks flow simulation. 2012.

32. Defense AA. Rotron PROPMAX 2 tubeaxial fans.

33. UJICAirfoil Coordinates Database. https://m-selig.ae.illinois.edu/ads/coord_database.html (accessed 12 February 2020).

34. Selig MS and Guglielmo JJ. High-lift low Reynolds number airfoil design. *J Aircr* 1997; 34: 72–79.

35. Król A and Król M. Study on numerical modeling of jet fans. *Tunneling Undergr Space Technol* 2018; 73: 222–235.

36. Çiğilengel YA and Cimbula JM. *Fluid mechanics: fundamentals and applications*, 2nd ed. Boston, MA: McGraw-Hill Higher Education, 2010.

37. White FM. *Fluid mechanics*. 8th ed. New Delhi, India: McGraw-Hill Education, 2017.

38. Cengel Y, Çengel YA and Cimbula JM. *Fluid mechanics: fundamentals and applications*. Boston, MA: McGraw-Hill Education, 2018.

39. Anderson JD Jr. *Fundamentals of aerodynamics*. 3rd ed. Boston, MA: McGraw-Hill Education, 2001.

40. Ziemiński JA and Whitlow JBB. NASA/industry advanced turboprop technology program. In: *16th congress of the international council of aeronautical sciences*, Jerusalem, Israel, 28 August–2 September 1988. Cleveland, OH: National Aeronautics and Space Administration.

41. Korpela SA. *Principles of turbomachinery*. Wiley Online Library, 2011.

42. Rogalsky T, Derksen R and Kocabiyik S. An aerodynamic design technique for optimizing fan blade spacing. In: *Proceedings of the 7th annual conference of the computational fluid dynamics society of Canada*, 1999, pp.2–29.

43. Lin S-C and Chou C-A. Blockage effect of axial-flow fans applied on heat sink assembly. *Appl Therm Eng* 2004; 24: 2375–2389.
44. Beri H and Yao Y. Effect of camber airfoil on self starting of vertical axis wind turbine. *J Environ Sci Tech* 2011; 4: 302–312.

45. Gheorghe A, Zolghadri A, Cieslak J, et al. Model-based approaches for fast and robust fault detection in an aircraft control surface servo loop: from theory to flight tests. *IEEE Control Syst Mag* 2013; 33: 20–84.

**Appendix**

**Notation**

| Symbol | Description |
|--------|-------------|
| $D_{hub}$ | diameter of the hub, $m$ |
| $D_{tip}$ | diameter of the tip, $m$ |
| $D_{tube}$ | diameter of the tube (i.e. computational domain), $m$ |
| $e_{ijk}$ | Levi-Civita function |
| $g$ | acceleration due to gravitational, $m/s^2$ |
| $h$ | thermal enthalpy, $J$ |
| $H$ | angular momentum, $kgm^2/s$ |
| $H_p$ | fluid head, $m$ |
| $k$ | turbulent kinetic energy, $J$ |
| $L$ | length of computation domain, $m$ |
| $M$ | molecular mass, $g/mol$ |
| $m$ | mass flow rate, $kg/s$ |
| $p$ | pressure, $Pa$ |
| $P_{rel}$ | relative pressure, $Pa$ |
| $P_s$ | pressure at sea level, $Pa$ |
| $r$ | distance from a point relative to the axis of rotation |
| $R$ | gas constant of air, $J/kgK$ |
| $S_i$ | mass distributed external force, $N$ |
| $T$ | temperature, $K$ |
| $T_m$ | torque of the fan, $Nm$ |
| $T_s$ | temperature at sea level, $K$ |
| $u_i$ | instantaneous fluid velocity, $m/s$ |
| $V$ | fluid velocity magnitude, $m/s$ |
| $x_i$ | coordinate direction |

**Greek letters**

| Symbol | Description |
|--------|-------------|
| $\gamma$ | specific heat ratio |
| $\varepsilon$ | turbulent dissipation, $J/kg$ |

| Symbol | Description |
|--------|-------------|
| $\kappa$ | thermal conductivity, $W/mK$ |
| $\rho$ | fluid density, $kg/m^3$ |
| $\tau_{i,j}$ | shear stress tensor, $N/m^2$ |
| $\tau_{i,j}^R$ | Reynolds shear tensor, $(m/s)^2$ |
| $\Omega$ | rotational speed, $rpm$ |

**Non-Dimensional Numbers**

| Symbol | Description |
|--------|-------------|
| $C_H$ | Coefficient of head |
| $C_p$ | Coefficient of power |
| $C_Q$ | Coefficient of capacity |
| $\gamma$ | specific heat ratio |
| $\eta$ | Fan efficiency |

**Subscripts and superscripts**

| Symbol | Description |
|--------|-------------|
| $H$ | head |
| $hub$ | diameter of the hub |
| $i,j,k$ | coordinate direction (i.e. $x$, $y$, $z$) |
| $m$ | magnitude of torque from CFD |
| $P$ | input power to the fan |
| $p$ | pressure head |
| $Q$ | capacity of volumetric flow rate |
| $R$ | Reynolds stress tensor |
| $rel$ | relative pressure |
| $rotation$ | rotational external source term modeling fan region |
| $s$ | condition at sea level |
| $tip$ | diameter of the tip of the fan |
| $tube$ | diameter of the tube |

**Acronyms and Abbreviations**

| Acronym | Description |
|---------|-------------|
| BEP | Best efficiency point |
| CAD | Computer-aided design |
| CFD | Computational fluid dynamics |
| EMA | Electromechanical actuator |
| FSP | fan static pressure |
| S1210 | Selig S1210 high lift low Reynolds number airfoil |
| SSCA07 | Sikorsky/Flemming SSC-A07 transonic rotorcraft airfoil |