Journal of Applied Fluid Mechanics, Vol. 18, No. 8, pp. 1953-1963, 2025.
Available online at www.jafmonline.net, ISSN 1735-3572, EISSN 1735-3645.
https://doi.org/10.47176/jafm.18.8.3325

High-efficiency Axial Flow Fan Design by Combining Through-flow Modeling, Optimization Algorithm and Computational Fluid Dynamics Simulation

C. Lee[1†], S. W. Kim[2], H. T. Byun[3] and S. H. Yang[4]

[1] *Department of Mechanical Engineering, University of Suwon, Hwaseong, Gyeonggi, 18323, Republic of Korea*
[2] *PIDOTECH, Songpa, Seoul, 05854, Republic of Korea*
[3] *KWTech, Seongnam, Gyeonggi, 13497, Republic of Korea*
[4] *Samwon E&B, Shiheung, Gyeonggi, 15078, Republic of Korea*

†*Corresponding Author Email:* clee@suwon.ac.kr

ABSTRACT

In this study, a high-efficiency axial fan design method and process are proposed by integrating the controlled blading design (CBD) method for the spanwise design of blade angles, the through-flow analysis method incorporating three-dimensional flow effects for performance prediction of the designed fan, and an optimization algorithm suitable for multi-variable problems. The main objective of this study is to obtain the optimal spanwise distribution of blade angles and chord length. To achieve this, the three-dimensional blade design process of the axial flow fan is established using the CBD model, in which the camber angle and setting angle along the spanwise direction are set as design variables, while the chord length along the spanwise direction is considered as another design variable. To predict the performance and efficiency of the designed fan, the through-flow analysis method is introduced, and the accuracy of flow and performance predictions using this method is verified by comparing with measurement results. A newly developed hybrid metaheuristic algorithm is applied as an optimization technique in the fan design and through-flow analysis program, enabling the optimal design of a high-efficiency axial flow fan. An optimization problem maximizing fan efficiency is defined and several design constraints are also set. The optimization algorithm is applied to the fan design and through-flow analysis program, achieving a very fast and simple optimization process and obtaining the optimal axial fan model. By comparing the optimal fan model with the initial fan model based on the free-vortex flow type, it is confirmed that fan efficiency is improved by 4.2 percentage points through this optimization. To verify the reliability of this optimization design method, CFD analysis, manufacturing, and testing are conducted for the optimized fan model. A comparison between the optimal design results and CFD calculation results demonstrates that this optimization method has very high predictive accuracy and design reliability. Furthermore, by comparing the design and CFD results of the optimized model with actual performance test results, the improvement in performance and efficiency through this optimization design method is validated. Additionally, the optimized axial fan derived in this study exhibits excellent performance characteristics, maintaining high efficiency and low power characteristics even under low flow conditions.

Article History

Received December 24, 2024
Revised March 9, 2025
Accepted April 9, 2025
Available online June 3, 2025

Keywords:

Axial flow fan
Blade angles
Computational fluid dynamics
Modeling and simulation
Design optimization method
Fan efficiency

1. INTRODUCTION

An axial fan is an essential machine for supplying air of adequate flow rate and pressure in many industrial, commercial, and residential facilities. In the modern industrial society, a large number of fans are used in various application fields, so it is reported that the power consumption of the fans is also very high (Benson, 2003). The carbon neutrality issue, which has recently attracted worldwide attention, requires high-efficiency design and operation of all machinery and facilities and in this context, design optimization technology for high efficiency,

NOMENCLATURE

c	chord length	ξ	stagger angle
I	rothalpy	μ	setting angle
i	incidence angle	θ_c	camber angle
m	meridional coordinate	ϕ	streamline slope angle
p	pressure	Ω	angular frequency of fan blade rotation
Q	entropy function	η_T	total efficiency (efficiency based on total pressure)
r	radial coordinate	**Subscript**	
T	temperature	1	blade inlet
V	flow velocity	2	blade outlet
X_i	design variable	h	hub
α	flow angle	m	mid-span or meridional
α'	blade angle	t	tip
δ	deviation angle		

inverter, and variable-pitch operation technology are actively developed and applied in the case of axial fans.

Determining the three-dimensional shape of fan blade is a priority for high-efficiency design of axial fans, and for this purpose, various fan design theories and methods have been proposed and applied. The well-known classical fan blade design methods include ways of determining the three-dimensional blade shape using the design concept of free vortex, forced vortex, or combined vortex (Dixon, 1998; Cohen et al., 2017). These methods have the advantage of being simple and easy to use, but there is a disadvantage that the effect of blade angle and curved surface along the blade span direction cannot be variously reflected in the fan design. The methods proposed to compensate for the shortcomings of these existing methods are controlled blading design methods, and through optimization of design variables such as camber angle, setting angle, and chord length of the blade section, aerodynamic factors such as lift coefficient, diffusion factor, or total pressure loss coefficient are controlled and adjusted to ultimately design a high-performance and high-efficiency fan. Wallis (1983) proposed a method of designing the blade of variable-pitch axial fan using the correlations between the flow angle and the blade section lift coefficient. Spuy and Backstrom (2002) obtained the optimal fan blade angle distribution to minimize the outlet kinetic energy of axial fan with variable-pitch rotor. Lee (2016) defined the camber angle, setting angle, and chord length of the blade section as design variables for axial fan design, and compared the performance characteristics of the fan according to different fan blade design methods. However, most of the aforementioned controlled blading design (CBD) methods are based on two-dimensional design theory and cascade experimental results, and the selection and control of aerodynamic factors depend on fan designer experience, so the concept and method of optimization can be applied to this CBD method for automating blade section designs and the three-dimensional blade formation, thereby obtaining excellent fan performance and efficiency (Kim et al., 2022; 2023).

The study of the optimal design of axial fans has been dealt with by many researchers thanks to recent advances in optimization techniques and breakthroughs in computer computing power. An important issue in such optimization research is how reliable the fluid mechanical design model can be constructed and how many key design parameters and constraints can be handled. The recently reported studies on the optimal design of axial fans mainly take the computational fluid dynamics model as the design model and combine optimization techniques with it. Samad and Kim (2009) proposed a surrogate-based optimization technique combined with the Reynolds stress Navier-Stokes equation Solver (RANS) method, which can be used in turbomachine's aerodynamic design. Bamberger and Carolus (2012) optimized the blade design of a forward-swept axial fan with a modified sweep strategy, using CFD simulation and acoustic measurement. Angelini et al. (2017) optimized the chord length and pitch angle distributions of axial fan rotor blades using Python code and a buff-force-search optimization algorithm, verified the optimization results using Open FOAM CFD computational simulation, and the CFD calculation results showed a 0.5dB reduction in fan noise level. Edward et al. (2021) performed the automated optimization process of designing a transonic fan blade using CFD code, DOE (Design of Experiment), and a surrogate model. The optimized fan blade showed that the overall pressure is increased by up to 24% at higher mass flow rates compared to the reference model. However, as can be seen from the aforementioned studies, most of these optimization studies focus on limited efficiency improvement or noise reduction, dealing with a small number of design variables and constraints due to the meshing complexity and long calculation time of the computational fluid dynamics model.

Therefore, this research presents a novel design tool—the CBD method and a through-flow model considering three-dimensional flow effects—for the optimal design of blade angle distribution along the spanwise direction of high-efficiency axial fans. By applying an optimization algorithm developed by the authors to this design program, the authors derived an optimal design for a high-efficiency fan and validated the results through CFD and experimental testing. Compared to existing CFD-based optimization studies, this research addresses a greater number of design variables and constraints, enabling reliable design optimization in a shorter time frame. It offers a more practical design system suitable for industrial applications.

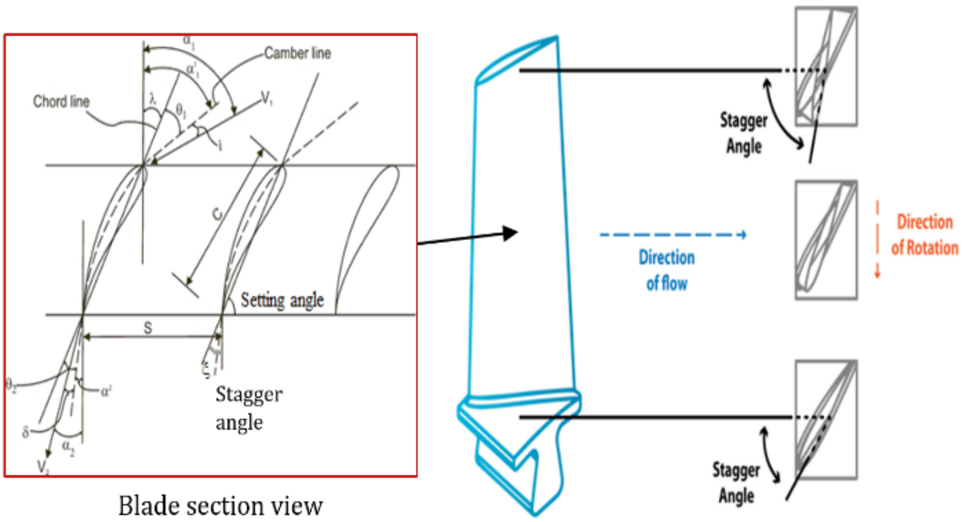

Fig. 1 Fan blade section and design parameters

2. FAN DESIGN AND PERFORMANCE PREDICTION METHODS

The fan design program used in this study is the FANDAS program (FANDAS, 2023) developed by the authors, and it is possible to design three-dimensional fan blade shape and predict performance accordingly. The introduction of the design method and performance prediction method of this study is as follows:

2.1 Fan Design Method

As shown in Fig. 1, the three-dimensional fan shape design is performed by designing blade sections and stacking the designed sections in the blade span direction (spanwise stacking). In this study, the controlled blading design (CBD) method is adopted for the blade section design, and the camber angle (θ_c), setting angle (μ), and chord length (c) are considered as design variables.

As can be seen in Fig. 1, given the camber angle, setting angle, and chord length, the angle at the leading edge (α_1') and the angle at the trailing edge (α_2') of the blade section are obtained from the following equations,

$$\theta_c = \alpha_1' - \alpha_2' \quad (1)$$

$$\mu = \frac{\pi}{2} - \xi \text{ , } \tan\xi = \frac{\tan\alpha_1' + \tan\alpha_2'}{2} \quad (2)$$

In this case, the stagger angle is represented as ξ , and for reference, the setting angle at the hub is the pitch angle of the blade. In addition, the incidence angle (i) at the leading edge of the blade is defined as follows

$$i = \alpha_1 - \alpha_1' \quad (3)$$

where α_1 means the flow angle at the inlet of the blade.

The camber line connecting the leading and trailing edges is obtained as single circular arc and the NACA6309 airfoil thickness distribution is put onto the camber line to complete the blade sectional design. This study focuses on optimizing the spanwise distributions of blade camber angle, setting angle, and chord length. Therefore, the camber line construction is simply based on the conventional circular arc method. The reason for using the NACA 6309 airfoil profile is that it has a wider incidence angle range with low-pressure loss compared to other airfoil profiles, making it suitable for the variable-pitch axial fan developed in this study.

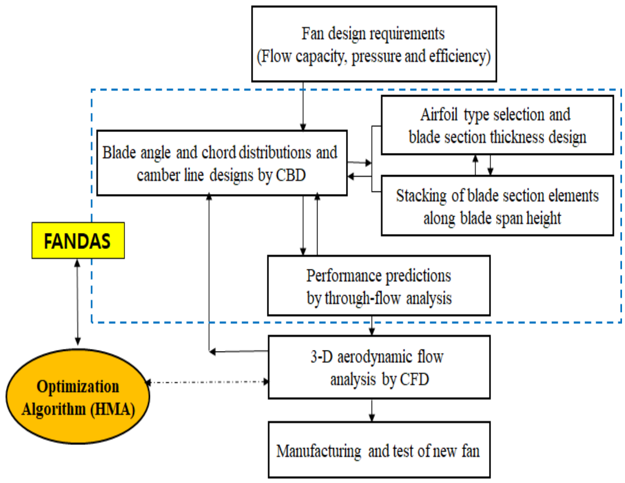

Fig. 2 Computation process of the design and through-flow analysis of axial fan

By stacking these designed sections in the blade span direction, a fan blade with three-dimensional curved surface is constructed. For reference, in this study, a three-dimensional fan blade is constructed by designing five blade sections in the span direction.

2.2 Fan Performance Prediction Method

The performance prediction method used in this study is a method of calculating the flow rate, pressure, efficiency, and power of the fan by mass-averaging the flow calculation results obtained by through-flow analysis on the streamline of air passing through the designed fan blades (see Fig. 2). This through-flow analysis method calculates the radial equilibrium equation for the meridional velocity (V_m) in Eq. (1),

$$\frac{dV_m^2}{dr} + A(r)V_m^2 = B(r) \quad (4)$$

$$\mathrm{A(r)} = 2sin^2\alpha[-\frac{sin\phi}{V_m}\frac{dV_m}{dm} + \frac{cos\phi}{r_m} + \frac{csc^2\alpha}{2}(\frac{1}{Q}\frac{dQ}{dr})$$

Table 1 Pressure loss and deviation models

Flow deviation and Loss source	Correlation models by
Deviation angle	Dixon (1998), Adams and Leonard (2005)
Profile loss	Koch and Smith (1976)
Secondary flow loss	Lee and Chung (1991)
Endwall flow loss	Horlock and Lakshminarayana (1970)
Tip clearance loss	Horlock and Lakshminarayana (1970) Simon and Leonard (2006)

$$+\frac{1}{2}\frac{d(cot^2\alpha)}{dr}+\frac{cot^2\alpha}{r}+\frac{2\Omega}{V_m}cot\alpha] \tag{5}$$

$$B(r)=2sin^2\alpha\left[\frac{1}{Q}\frac{d(IQ)}{dr}+\frac{\Omega^2r^2}{2}\left(\frac{1}{Q}\frac{dQ}{dr}\right)\right] \tag{6}$$

$$Q=\left(\frac{p_{02}}{p_{01}}\right)^{\frac{\gamma-1}{\gamma}}/\left(\frac{T_{02}}{T_{01}}\right) \tag{7}$$

Here the flow angle and total pressure loss of air passing through the blades are obtained by the correlation models in Table 1 and reflected in the entropy function (Q) calculation.

Through-flow analysis method is a critical tool in turbomachinery design, used to predict overall aerodynamic performance and optimize blade shape before conducting more detailed 3D CFD simulations. Recent developments in through-flow models focus on integration with 3D flow effects (Fan et al., 2024). For this reason, the authors' method also introduces deviation angle and pressure loss models to account for 3D flow effects (refer to Table 1). The deviation angle model reflects changes in the flow angle caused by three-dimensional flow effects near the hub and tip.

The profile loss model is applied along each streamline, the secondary flow model is applied as a parabolic function in the span direction centering on the mid-span, and the end wall and tip clearance loss models are applied only to tip and hub streamlines. The three-dimensional flow effect inside the fan blades can be reflected through the spanwise pressure loss distribution models. Detailed mathematical expressions and explanations for the method and models for the through-flow analysis of this study are described in detail in reference (Lee, 2021).

2.3 Fan Flow and Performance Prediction Results

Figure 3 shows the comparison between the prediction and measurement results for the distribution of flow velocities inside the low pressure compressor. The prediction results by this method agree well with the measurement results (Britsch et al., 1979) in the radial direction of the compressor blade span, and the comparison results near the hub and tip with remarkable three-dimensional flow effect are also very positive. Figure 4 shows the total pressure loss coefficient distribution along the blade radius. The through-flow analysis results agree well with the measured data, particularly demonstrating relatively reliable predictions of pressure loss near the hub and tip, where 3D flow effects are significant.

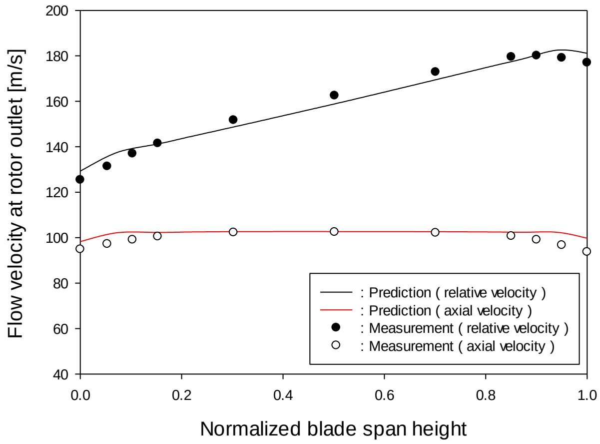

Fig. 3 Velocity distributions of axial compressor

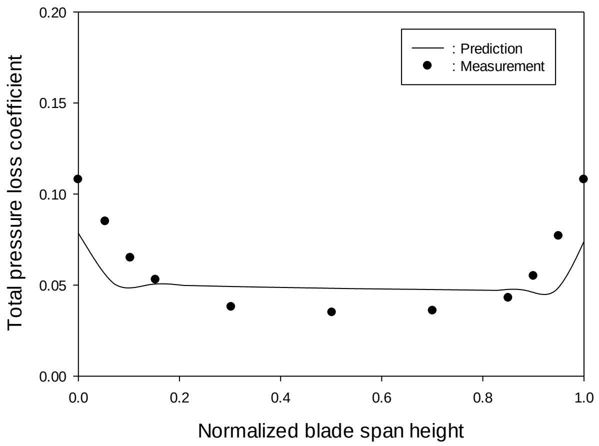

Fig. 4 Pressure loss distribution of axial compressor

Figure 5 compares the efficiency curves predicted for two fans designed by the free vortex design (FVD) or the combined vortex design (CVD) method with the CFD results (Lee, 2016), and the prediction results by this method agree well with the CFD results in the entire flow range. Figure 6 shows a comparison that matches the prediction results and test results of the pressure curves under the variable operating conditions of the general application fan designed using the controlled blading design (CBD) method (Spuy, 1997). As previously shown in the results of Figs. 4, 5 and 6, it can be seen that this

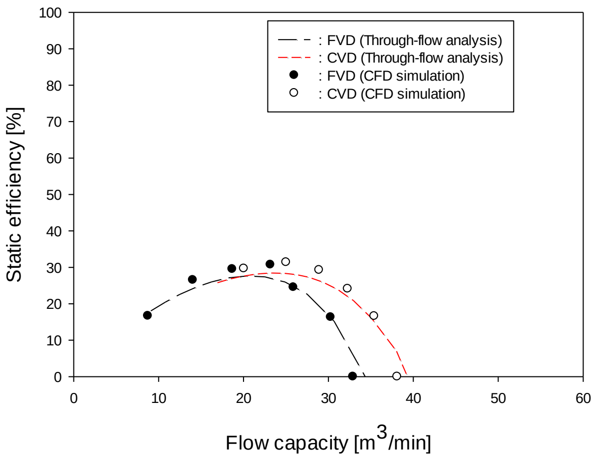

Fig. 5 Efficiency curves of axial flow fans

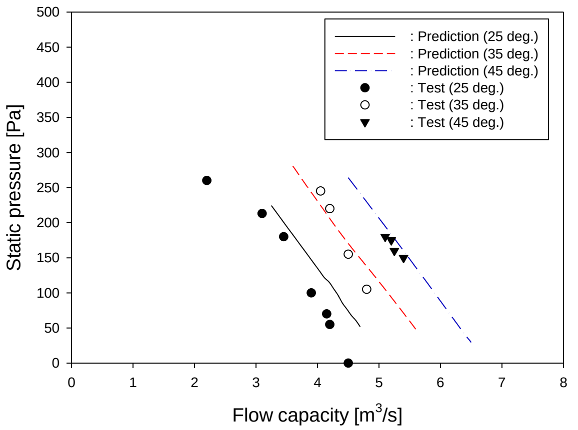

Fig. 6 Pressure curves of axial flow fan

through-flow analysis method is suitable for performance prediction of fans designed with different blade design specifications and methods.

3. Fan Design Optimization Method, Results and Verification

In this study, the optimal design problem is constructed using the axial fan design program mentioned above, and the optimization method, and then the optimal design results are obtained.

3.1 Optimization Problem and Method

The purpose of this optimization is to maximize the efficiency of the axial fan blade, and 14 design parameters are dealt with for the camber angle, setting angle, and chord length at different blade span locations and number of blade blades as the blade design variables. Given the design variables for camber angle, setting angle, and chord length, the fan blade is designed using the controlled blading design (CBD) method of the FANDAS program (refer to section 2.1). In addition, practical design constraints for fan design are also reflected in the optimization problem.

The objective function and design variables of this optimization problem are as follows:

$$\text{Maximize} \quad \eta_T = f(X_1, X_2 \dots\dots X_{13}, X_{14}) \tag{8}$$

where X_1, X_2, X_3, X_4, X_5: the camber angles at the 0, 25, 50, 75, 100% span locations, X_6, X_7, X_8 X_9, X_{10}: the setting angles at the 0, 25, 50, 75, 100% span locations, X_{11}, X_{12}, X_{13} : the chord lengths at the 0% (hub), 50% (mid-span), and 100% (tip) span locations, and X_{14} : the number of blades (discrete variable).

Fourteen design constraints used in this optimization are summarized in Eqs. (9)- (13),

$$550 < p_T < 700\ [Pa] \tag{9}$$

$$0 < X_1, X_2 \dots, X_{10} < 90\ [deg.] \tag{10}$$

$$0.3 < \frac{X_{11}}{\left(2\pi r_h / X_{14}\right)} < 2.5 \tag{11}$$

$$0.3 < \frac{X_{12}}{\left(2\pi r_m / X_{14}\right)} < 2.5 \tag{12}$$

$$0.3 < \frac{X_{13}}{\left(2\pi r_t / X_{14}\right)} < 2.5 \tag{13}$$

where r_t and r_h are 0.76m and 0.25m respectively, and the rotation speed of the fan is 1200 rpm. Eq. (9) defines the permissible range for the design pressure of the fan in this optimization. Eq. (10) specifies the ranges for the design variables, camber angle and setting angle. Eqs. (11) through (13) present the design ranges for solidity at the blade's hub, mid-span, and tip locations. These ranges are based on those commonly applied to low-speed fans and high-speed compressors (Lewis, 1996; Dixon, 1998;).

To solve the aforementioned optimization problem, the FANDAS program is used to design the fan blade shape and to predict the performance of the fan blade, and the optimal design solution is obtained by applying the optimization technique of Hybrid Metaheuristic Algorithm (HMA) to the FANDAS design/prediction results. HMA combines two different metaheuristic algorithms, differential evolution (DE) and cuckoo search (CS), using bi-population concepts (Kim, 2022; PIANO, 2023). The description of the optimization technique used in this study is described in detail in references (Kim, 2022; PIANO, 2023).

In general, optimization algorithms can be broadly classified into global search and local search methods. Compared to local search methods, global search methods have a much higher probability of finding the global solution. However, they typically require a significantly larger number of evaluations. Therefore, when each evaluation takes a long time, directly integrating a global search method with simulation software can be impractical due to time constraints. However, in the case of the FANDAS as design tool, since the evaluation time per analysis is extremely short, the time burden of using a global search method is significantly reduced. Consequently, this study employs the HMA algorithm with a global search method capable of finding the global optimum.

The HMA used in this study also offers several advantages over the adjoint-based method and Genetic

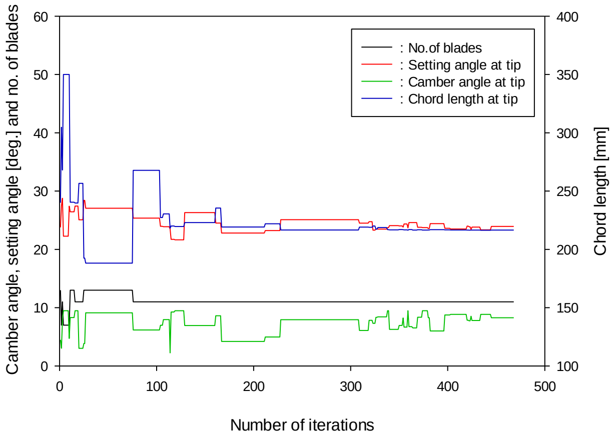

Fig. 7 Convergence histories of design variables

Table 2 Design optimization results

Design variable	Initial design	Optimal design
X_1 [deg.]	14.7	17.9
X_2 [deg.]	9.0	13.5
X_3 [deg.]	6.4	11.4
X_4 [deg.]	5.1	8.7
X_5 [deg.]	4.5	8.3
X_6 [deg.]	49.0	53.8
X_7 [deg.]	38.7	42.2
X_8 [deg.]	31.9	33.4
X_9 [deg.]	26.7	27.3
X_{10} [deg.]	23.9	23.9
X_{11} [mm]	240	150
X_{12} [mm]	240	162
X_{13} [mm]	240	219
X_{14} [ea]	13	11

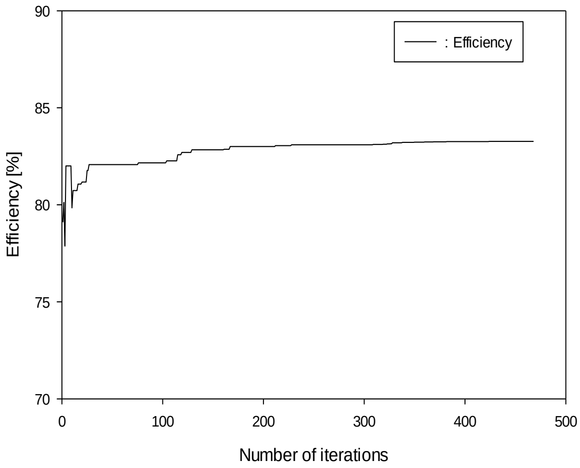

Fig. 8 Convergence history of objective function

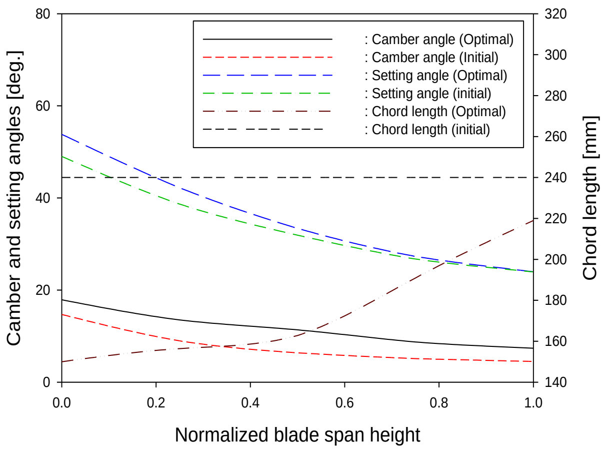

Fig. 9 Camber and setting angle distributions of the optimal fan blade

Algorithm (GA), commonly used with CFD software, in terms of search capability and convergence speed. The adjoint-based method relies on sensitivity derivative information, which may limit its effectiveness in highly nonlinear problems or cases with multiple local optima, but the HMA combines multiple search strategies of DE and CS to increase the chances of finding the global optimum in complex, nonlinear problems. And the HMA can handle non-differentiable functions and discrete variables. GA has strong global search capabilities, but its randomness can result in slower convergence. The HMA, by combining global and local search techniques, provides faster convergence and better solutions (Pulliam et al., 2003).

This optimization problem requires a convergence process of design variables, design constraints, and objective functions through iterative calculations to obtain an optimal solution. As can be seen in Figs. 7 and 8, design variables and objective function are converging robustly, and all design constraints are not violated. And this optimization process shows relatively short computation time and rapid convergence characteristics compared to similar CFD-based optimization studies (Samad & Kim, 2009; Choi et al., 2023).

3.2 Optimal Design Results

The optimal design variables are summarized in Table 2, and the spanwise distributions of the design variables (camber angle, setting angle and chord length) obtained through optimization are shown in Fig. 7. The number of the blades (X_{14}) is optimized to be 11. For reference, the initial design, which is the comparison target of the optimal design, is the case where it is designed by free vortex design (FVD) method under the condition of total pressure of 590 Pa.

As can be seen in Fig. 9, the optimal camber angle tends to be larger over the entire span compared to the initial design, and this result means that the optimal design is designed with higher pressure than the initial design. The optimal setting angle is somewhat higher in the region from the hub to the mid-span when it is compared to the initial setting angle. And the optimal chord length tends to increase from the hub to the tip compared to the initial design with constant chord length.

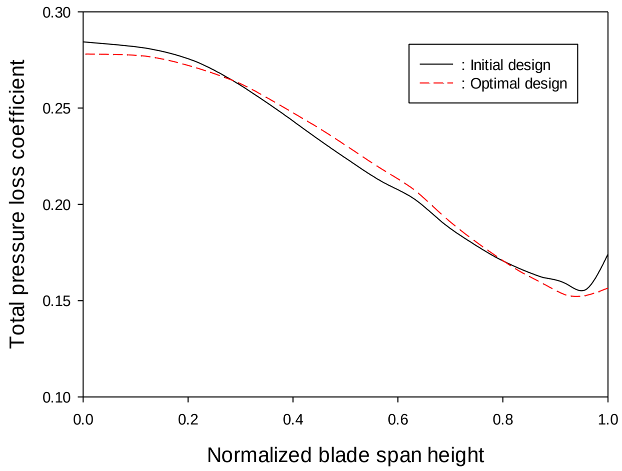

Fig. 10 Total pressure loss coefficient distribution of the optimal fan blade

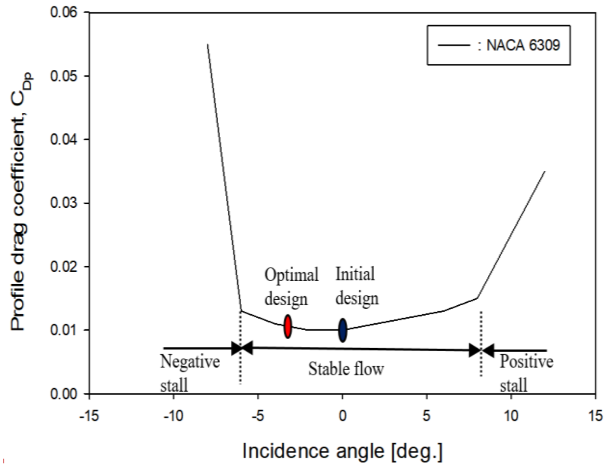

Fig. 12 Design incidence angle comparison between the optimal and the initial fan designs

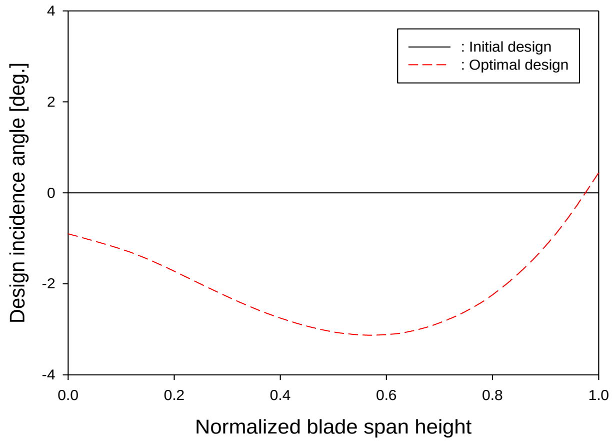

Fig. 11 Design incidence angle distribution of the optimal fan blade

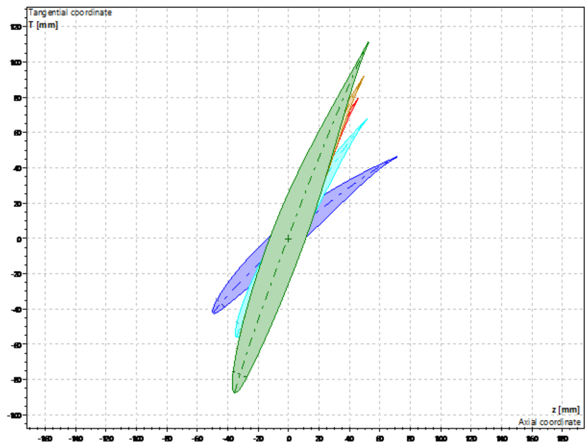

Fig. 13 Section designs of the optimal fan blade

Figure 10 shows the comparison of the total pressure loss coefficient between the optimal design and the initial design. The total pressure loss coefficient of the optimal fan blade is significantly lower near the tip and hub than the initial design, while it is somewhat higher near the mid-span. Since the total pressure loss is obtained by multiplying the pressure loss coefficient by the inlet dynamic pressure and the dynamic pressure at the tip is relatively greater than that of the mid-span, the total pressure loss at the optimal blade tip is significantly reduced and the overall total pressure loss is minimized compared to the initial design even if the total pressure loss increases slightly at the mid-span. This comparison result indicates an improvement in the efficiency of the optimal blade. In addition, these results demonstrate that this optimization process automatically designs and combines the blade sections along span to minimize the overall total pressure loss of the fan blade.

The distribution of the design incidence angle at the leading edge of the blade of this optimal design is shown in Fig. 11. Compared to the initial design with constant design incidence angle of 0 deg. over entire blade span, it can be seen that the optimal design is designed to have design incidence angle of -3 deg. in the mid-span region. This incidence angle result in the mid-span is explained in connection with the result of slight increase in the total pressure loss in the mid-span of Fig. 10.

Figure 12 also compares the design incidence angles in the mid-span of the optimal and initial design on the profile drag coefficient curve of the NACA 6309 airfoil used in the fan design (Eastman et al., 1935). As can be seen in Fig. 13, the optimal design has an incidence angle range of about 11 deg. from the design point to the positive stall limit, and in this range, relatively low pressure loss and stable flow are secured, so reducing the flow rate of the optimal fan model (or increasing the incidence angle) is expected to maintain higher efficiency than the initial design, and phenomena such as stall and surge are also expected to occur under lower flow conditions.

Figures 13 and 14 show the five section designs and three-dimensional shapes of the optimal fan blade by spanwise stacking of the section designs, and Table 3 compares the performance parameters of the optimal fan blade with the initial design. As can be seen in Table 3, the total pressure and efficiency of the fan obtained through the optimal

Table 3 Performance data of the optimal fan blade

Parameter	Initial design	Optimal design
Flow rate [m^3/min]	3,300	3,300
Total pressure [Pa]	590	689
Efficiency [%]	79.1	83.3
Power [kW]	41.0	45.3

Table 4 Performance data of the optimal fan blade

Parameter	Optimal Design	CFD Calculation	Relative Error [%]
Efficiency [%]	83.3	80.9	+3.0
Power [kW]	45.3	49.6	-8.7
Total pressure [Pa]	689	760	-9.3

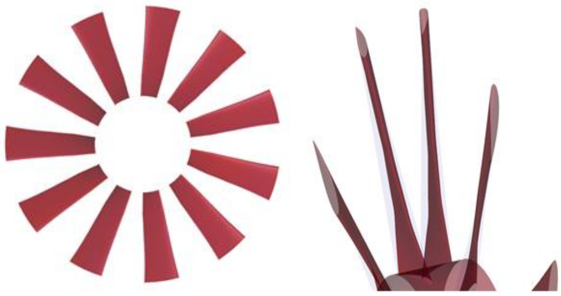

Fig. 14 3-D shape of the optimal fan blade

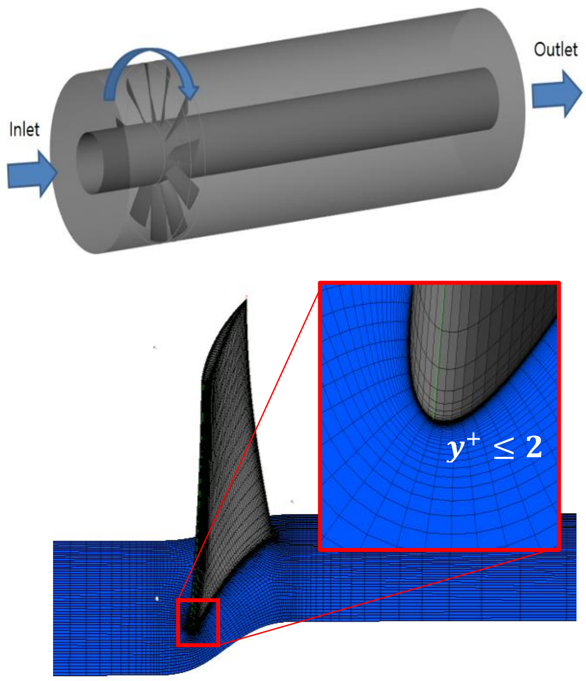

Fig. 15 Mesh system for the optimal fan blade

design are increased by 99 Pa and 4.2% compared to the initial design.

3.3 Verification of the Optimal Design

CFD modeling and numerical simulation are performed on the optimal fan blade to verify the design result of the optimal fan blade obtained in this study. The ANSYS CFX code is used for CFD calculations, and

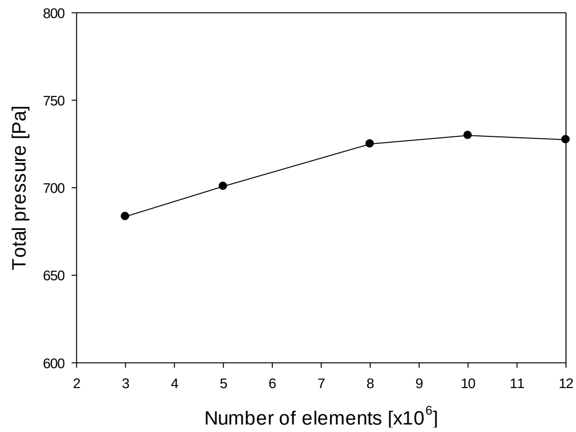

Fig. 16 Mesh dependency test

frozen rotor scheme and $\kappa - \omega$ SST turbulence model are adopted (ANSYS, 2023). This study utilizes steady-state RANS (Reynolds-stress Averaged Navier-Stokes equations) solver. For the discretization scheme, the 2nd-order upwind method is applied to velocity, while the 1st-order upwind method is used for pressure, turbulent kinetic energy, and turbulent energy dissipation rate. For the boundary conditions, a specified mass flow condition is set at the outlet, while a specified total pressure condition is applied at the inlet.

Figure 15 shows the computation domain and mesh system of the CFD model used in this study, and the mesh system uses structured cells and the number of elements is selected as about 8 million after mesh dependency test of Fig. 16. Body-fitted mesh inflation is used for mesh refinement near blade surfaces, tip clearance regions, and leading/trailing edges (refer to Fig. 15). The cells near the wall are generated with a fine resolution to achieve $y^+<2$ for accurate viscous boundary layer analysis. The wall cells have a high aspect ratio due to y^+ level on the wall and stretching around the blade outlet. However, the prediction error due to different mesh resolutions caused by aspect ratio remains within 0.51%.

Table 4 shows the comparison between the optimal design and the CFD calculation results, which means the

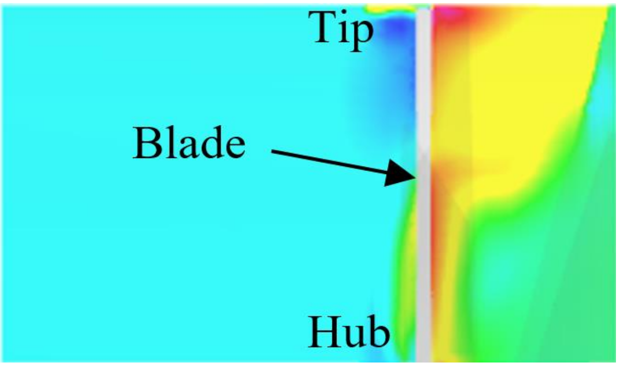

Fig. 17 Total pressure distribution of air passing through the optimal fan blade

Fig. 18 Assembly of manufactured optimal fan blades

present optimal design method is very reliable in high efficiency fan design within 1.3 percent of relative error.

Figure 17 shows the distribution of the total pressure of the air passing through the optimal blade. The total pressure of the air is increasing while passing through the blade, and it seems that no pressure drop due to flow separation occurs in the entire span. In particular, the increase in total pressure is remarkably large near the tip, and this result is due to the decrease in the total pressure loss coefficient near the tip by the optimal design in Fig. 8.

The optimal fan blades are manufactured and assembled with outlet guide vanes (OGV), and then tested in the Samwon E&B test facility according to ISO standard (refer to Fig. 18). And the existing fan model of the Samwon E&B is also compared with the optimal design.

Figures 19, 20 and 21 show the total pressure, efficiency, and power curves of the optimal fan blade. The results predicted by the CFD method for the optimal blade agree very well with the optimum design point and the test results are also favorably compared with the optimum design, and from these results, the reliability of the optimal design method of this study can be confirmed. In addition, as shown in Fig. 20, the optimal fan blade maintains relatively high efficiency and low power under flow

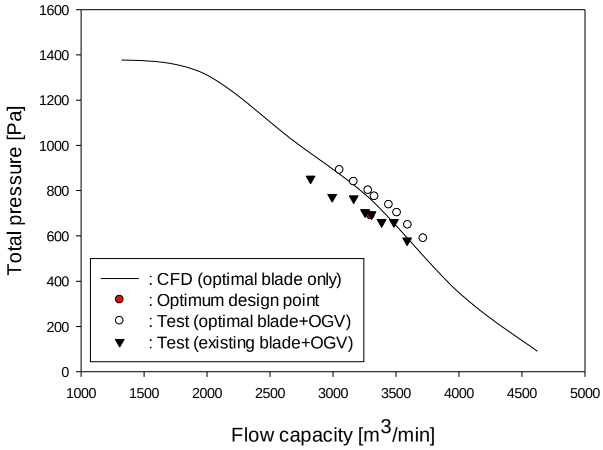

Fig. 19 Total pressure curve of the optimal fan blade

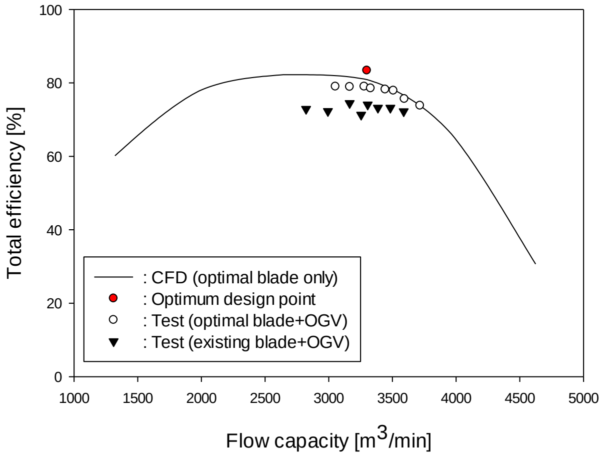

Fig. 20 Efficiency curve of the optimal fan blade

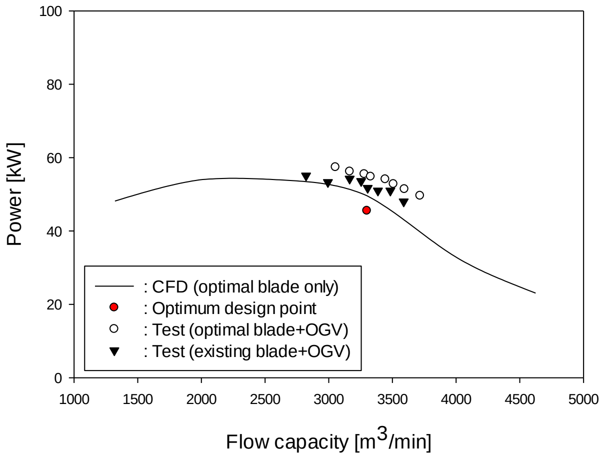

Fig. 21 Power curve of the optimal fan blade

conditions lower than the design point as expected in Fig. 10.

Based on the research results of this study, it is found that the proposed design program and optimization technique are effectively applied to the optimization of the camber angle, setting angle, and chord length distributions along the blade span of the fan blade. However, since this optimization method is based on through-flow modeling, it has limitations in addressing the effects of design factors related to highly complex and localized three-dimensional flows and vortices inside the fan blade (e.g., clearance size, wake interaction) on fan efficiency. Therefore, for design problems involving such complex three-dimensional flow fields, a CFD-based optimization approach should be used, while the proposed method of this study can be utilized in parallel for optimizing the basic blade shape. This combined approach is expected to enhance design productivity in the fan industry.

4. Conclusion

This study presents an optimal design method and process for the development of high-efficiency axial flow fan. The design program is constructed so that the three-dimensional blade design of the axial fan is conducted by performing the blade section designs with CBD method and stacking them along blade span direction, and the performance prediction of the designed fan is made through the through-flow analysis. By combining the design program and a HMA optimization algorithm proposed in this study, the optimal camber angle, setting angle, and chord length distributions of the fan blade are obtained, and the optimal blade shows an efficiency improvement of 4.2% compared to the initial design. Furthermore, in this study, the reliability of the optimal design results is confirmed through test, CFD modeling and numerical simulation, and the test and CFD results show that the optimal fan blade maintain high efficiency and low power over entire flow range.

Acknowledgements

This work was supported by the Korea Institute of Energy Technology Evaluation and Planning (KETEP) grant funded by the Korea government Ministry of Trade, Industry & Energy (MOTIE), Republic of Korea. (No. 2021202080026B).

Conflict of Interest

The authors declare that they do not have any conflict of interest.

Authors Contribution

C. Lee: Conceptualization, Investigation, Resources, Data curation, Writing – original draft, Visualization, and Supervision. **S. W. Kim** and **H. T. Byun:** Validation, Data curation, Visualization. **S. H. Yang:** Writing – review & editing, Visualization. All authors have read and agreed to the published version of the manuscript.

References

Adams, O., & Leonard, O. (2005). *A Quasi-one dimensional model for axial compressor.* ISABE proceedings. https://hdl.handle.net/2268/20889

Angelini, G., Bonanni, T., Corsini, A., Delibra, G., Tieghi, L., & Volponi, D. (2017). Optimization of an axial fan for air cooled condensers. *Energy Procedia*, *126*, 754-761. https://doi.org/10.1016/j.egypro.2017.08.236

ANSYS CFX user manual. (2023). ANSYS. https://www.ansys.com

Bamberger, K., & Carolus, T. (2012). Optimization of axial fans with highly swept blades with respect to losses and noise reduction. *Noise Control Engineering Journal*, *60*(6), 716-725. https://doi.org/10.3397/1.3701043

Benson, G. (2003). *Improving Fan System Efficiency*, US Department of Energy. https://docs.nrel.gov/docs/fy03osti/29166.pdf

Britsch, W. R., Osborn, W. M., & Laessig, M. R. (1979). *Effects of diffusion factor, aspect ratio, and solidity on overall performance of 14 compressor middle stages.* NASA TP-1523.

Choi, B. Y., Lee, H. J., Kim, S. W., Choi, D. H., Lee, C., & Yang, S. H. (2023). *Optimization of axial fan with variable pitch using AADO technique*. KSFM Winter Conference Proceedings. https://www.dbpia.co.kr/journal/articleDetail?nodeId=NODE11641361

Cohen, H., Rogers, G. F. C., Stranznicky, R., Saravamuttoo, H. I. H., & Nix, A. (2017). *Gas Turbine Theory*, Pearson.

Dixon, S. L. (1998). *Fluid Mechanics and Thermodynamics of Turbomachinery*, 4th edition, Buterworth & Heinemann.

Eastman, N. J., Kenneth, E. W., & Robert, M. P. (1935). *The characteristics of 78 related airfoil sections from tests in the variable-density wind tunnel*. NACA report No. 460.

Edward, D. R., Fanny, B. C., & Jean, C. B. (2021). *Optimization of a high pressure industrial fan.* ASME Turbo Expo 2021 proceedings. https://doi.org/10.1115/GT2021-58967

Fan, L., Xiawen, Z., Yaping, J., & Chuhua, Z. (2024). *Automatic and rapid throughflow-based optimal design method for multi-stage axial-flow compressors*. Springer.

FANDAS user manual, (2023). KWT solutions. http://www.kwtsolutions.com

Horlock, J. H., & Lakshminarayana, B. (1970). Secondary flows: Theory, experiment and applications in turbomachinery aerodynamics. *Annual Review of Fluid Mechanics*, *5*, 247-280.

Kim, S. W., Choi, B. I., Choi, D. H., Lee, C., & Yang, S. H. (2022). A study on a screening method for dimensionality reduction of large fluid machine optimization problems. *The KSFM Journal of Fluid Machinery*, *25*, 48-54. https://doi.org/10.5293/kfma.2022.25.3.048

Kim, S. W., Choi, B. I., Choi, D. H., Lee, C., & Yang, S. H. (2023). Multi-objective design optimization and data analysis considering the efficiency and variable environment of an axial fan. *The KSFM Journal of Fluid Machinery*, *26*, 143-151. https://doi.org/10.5293/kfma.2023.26.6.143

Koch, C. C., & Smith Jr., L. H. (1976). Loss sources and magnitudes in axial-flow compressors. *Journal of Engineering for Gas Turbines and Power*, *98*, 411-424. https://doi.org/10.1115/1.3446202

Lee, C. (2016). *Integrated computer program for design, performance and noise analyses of automotive cooling axial flow fans*. Technical report, Hanon systems.

Lee, C. (2021). A performance prediction method of the axial flow fans with blade sweep. *The KSFM Journal of Fluid Machinery*, *24*, 24-29. https://doi.org/10.5293/kfma.2021.24.5.024

Lee, C., & Chung, M. K. (1991). Secondary flow loss and deviation models for through-flow analysis of axial flow turbomachinery. *Mechanics Research Communications*, 403-408. https://doi.org/10.1016/0093-6413(91)90053-Y

Lewis, R. I. (1996). *Turbomachinery Performance Analysis*, ARNOLD

PIAnO User manual, (2023). PIDOTEC. https://www.pidotech.com.

Pulliam, T. H., Nemec, M., Holst, T., Zingg, D. W., & Kwak, Dochan. (2003). *Comparison of evolutionary (Genetic) algorithm and adjoint methods for multi-objective viscous airfoil optimizations*. AIAA paper 2003-0298. https://arc.aiaa.org/doi/abs/10.2514/6.2003-298

Samad, A., & Kim, K. Y. (2009). Surrogate based optimization techniques for aerodynamic design of turbomachinery. *International Journal of Fluid Machinery and Systems*, *2*, 179-188. https://doi.org/10.5293/IJFMS.2009.2.2.179

Simon, J. F., & Leonard, O. (2006). Modeling of 3-D losses and deviations in a through flow analysis tool. *Journal of Thermal Science*, *16*, 1-7. https://orbi.uliege.be/bitstream/2268/20885/1/Paper-2.pdf

Spuy, J. (1997). *The design of a low-noise rotor-only axial flow fan series* [Master of Engineering Thesis, Department of Mechanical Engineering, University of Stellenbosch]. http://hdl.handle.net/10019.1/20977

Spuy, S. J., & Backstorm, T. W. (2002). Performance of rotor-only axial fans designed for minimum exit kinetic energy. *R & D Journal*, 18, 63-69. http://hdl.handle.net/10019.1/75248

Wallis, R. A. (1983). *Axial Flow Fans and Ducts*, 2nd edition, John Wiley & Sons.