*Conference on Modelling Fluid Flow (CMFF'22)*
*The 18th International Conference on Fluid Flow Technologies*
*Budapest, Hungary, August 30 -September 2, 2022*

# PERFORMANCE PREDICTION AND CFD ANALYSES OF A VARIABLE-PITCH AXIAL FLOW FAN

Chan LEE[1], Hyun Gwon KIL[2], Eui Jong NOH[3] , Sang Ho YANG[4]

[1] Corresponding Author. Department of Mechanical Engineering, Faculty of Mechanical Engineering, University of Suwon. Hwaseong, South Korea, Tel.: +82 31 220 2219, E-mail: clee@suwon.ac.kr
[2] Department of Mechanical Engineering, Faculty of Mechanical Engineering, University of Suwon. E-mail: hgkil@suwon.ac.kr
[3] Department of Mechanical Engineering, Student, University of Suwon.
[4] Research Lab., Vice President, Samwon E&B, Siheung, South Korea

## ABSTRACT

Axial flow fans have been widely used for the air-conditioning and the ventilation systems of industrial, commercial and residential sectors, and their performance and efficiency are being improved by variable-pitch operation. The present paper investigates the effect of three dimensional blading design method of axial flow fan on fan performance and efficiency under variable-pitch operation. The three dimensional blade geometry of an axial flow fan is designed by free vortex, combined vortex or controlled blading design method. The three fan models designed by the different blading design methods are examined on the variable-pitch fan operations by comparing their performance predictions by streamline curvature method with the flow field analyses by CFD technique and measurement by chamber-type test facility of a existing fan model. From the comparison results, combined vortex design method is superior to the other competing desugn methods in the aspect of wide range of flow capacity while free vortex design or controlling blading design method is more proper to improve fan efficiency.

**Keywords: Axial flow fan, CFD modeling, Fan blading design method, Fan performance prediction, Variable-pitch fan operation**

## NOMENCLATURE

| | | |
|---|---|---|
| $C_L$ | [-] | section lift coefficient |
| $p_T$ | [*Pa*] | total pressure |
| $r$ | [m] | radial distance |
| $\underline{V}$ | [*m/s*] | absolute flow velocity |
| $\alpha$ | [*deg.*] | absolute flow angle |
| $\sigma$ | [-] | solidity |
| $\rho$ | [$kg/m^3$] | density of air |
| $\Omega$ | [-] | angular frequency |

**Subscripts and Superscripts**

| | |
|---|---|
| m | mean value |
| 1, 2 | blade inlet, blade outlet |
| $\theta$ | tangential direction |
| ' | blade design value |

## 1. INTRODUCTION

As fans are important elements of air conditioning and ventilation systems in industrial, commercial, and residential sectors, it is very urgent and essential to reduce the power consumption of fans in each field. For this reason, recently, there is a need to develop the fan capable of operating in wide range and with high efficiency, and fan rotation speed control method using inverter has already been applied and variable-pitch fan operation method is being actively applied to axial fan application systems. Many researches have been being conducted on the designs and analyses of variable-pitch fans for achieving high efficiency over wide flow capacity range[1]. Wallis proposed a fan blading design method of variable-pitch fan by using the correlations of flow angle and section lift coefficient[2]. According to the Howden fan development study[3], the determination of spanwise blade angle distribution is very critical design issue of variable-pitch fan and should be optimized through applying various design methods and verifying their effects by measurement and CFD modeling. Spuy and Backstrom[4] also applied an optimization procedure to minimize exit kinetic energy of variable-pitch rotor-only axial fan, and then designed optimal spanwise fan blade angle distribution.

Thus, this paper conducted design studies and flow analyses, performance predictions for examining the effects of different 3D blading design methods of axial fan on fan performance and efficiency during variable-pitch operation. Furthermore, based on the selelcted appropriate blading design method, an design optimization is conducted for maximizing total efficicny of variable-pitch fan and the optimal fan is verified by CFD method.

## 2. FAN DESIGN AND PERFORMANCE PREDICTION METHOD

In the case of variable-pitch fan, there is the remarkable change in incidence angle of air flow due to the change in setting angle of varible-pitch rotor blade. Since the flow rate, pressure, and efficiency of fan are significantly affected due to this incidence angle change, in order to ensure wide flow range and high efficiency of variable-pitch fan, it is necessary to design rotor blade sections to effectively respond to the changes in incidence angle[5].

Therefore, using the FANDAS code as Korean fan design program, the present study employs three different blading design methods, free vortex design(FVD), combined vortex design (CVD) and controlled blading design (CBD), to design and stack blade sections of variable-pitch fan. Table 1 summarizes the design rules of the three design methods to determine the tangential velocity distribution and then the blade angles of fan rotor blade outlet[2,5,6].

Table 1 Design rules of three design methods

| Design Method | Design rule | Remark |
|---|---|---|
| Free Vortex Design (FVD) | $V_{\theta 2} = \frac{a}{r}$ | $\frac{\Delta p_T}{\rho\Omega} = a$ |
| Combined Vortex Design(CVD) | $V_{\theta 2} = \frac{a}{r} + br$ | $\frac{\Delta p_T}{\rho\Omega} = a + br_m^2$ |
| Controlled Blading Design(CBD) | Adjust and control the spanwise distributions of blade stagger and camber angles of $\xi(r)$ and $\theta c(r)$. | Wallis(1981), Mckenzie(1997): $C_L$ is given on each blade section, and then the corresponding $\alpha_1'$ and $\alpha_2'$ are determined $C_L = \frac{2cos\alpha_m}{\sigma}(tan\alpha_1 - tan\alpha_2)$ |

Before the application of the FANDAS code to new variable-pitch fan design, the reliability and prediction accuracy in the design and performance prediction of the FANDAS code are verified. First verification examples are three automotive cooling fans which are designed by different blading design methods of the FANDAS code. The rotation speed of the three fan models is 1940 rpm, and the chord length of the fans varies from 0.049 m (hub) to 0.078 m (tip), and the number of blades is 7[8]. The shapes of the three fan blades are shown and compared in Fig. 1. For the CFD modeling of three fan models, unstructured mesh systems on chamber type flow domain are constructed with about 1 million elements and high mesh quality. On the mesh system shown in Fig. 2, numerical calculations are conducted on the three fan models by the ANSYS CFX code[9] with frozen rotor scheme and k-ω SST turbulence models. The airflow passing over the blade surface calculated by the CFD technique is also depicted in Fig. 2. Performance predictions by the CFD simulations are conducted on three fan models designed by the FVD, the CVD and the CBD methods. In addition, for a fan model designed by the CBD method, a performance test is performed in an outlet chamber set-up facility with multiple nozzles according to AMCA standard.

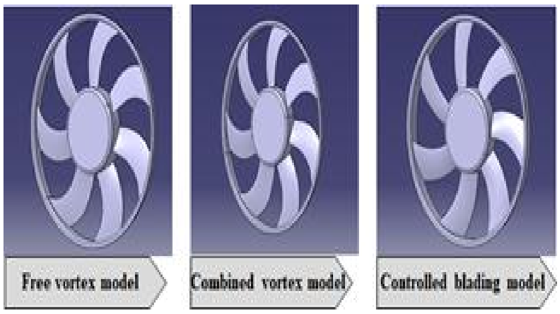


Fig. 1 Three different fan design models

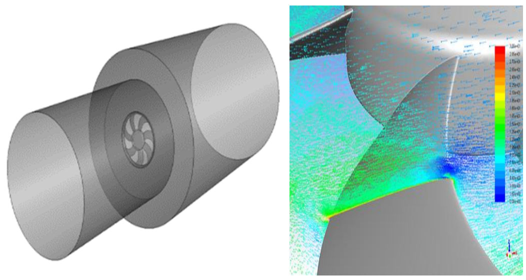


Fig. 2 Computational domain in fan

Fig. 3 shows the comparison between the static pressure curves by the FANDAS code prediction, the CFD calculation and the measurement results. It can be seen in Fig. 3 that the CFD calculation results for the CBD fan are well agreed with the measurement results and the prediction accuracy of the present CFD method is excellent. Fig. 3 also shows the FANDAS prediction, the CFD calculation and the measurement results are well agreed over the entire flow range of the FVD, the CVD and the CBD fans.

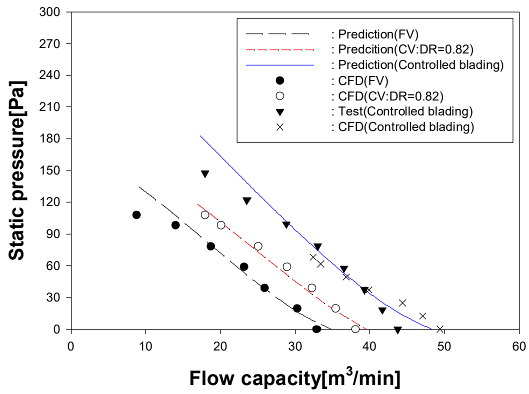


Fig. 3 Static pressure curves of three fan models

Fig. 4 also shows that the efficiency prediction results of the FANDAS code are relatively well consistent with the CFD and the measurement results. From these results of Figs. 3 and 4, it is judged that the FANDAS code is very suitable for designing 3D fan blade shapes and predicting the fan performance regardless of fan blade design method.

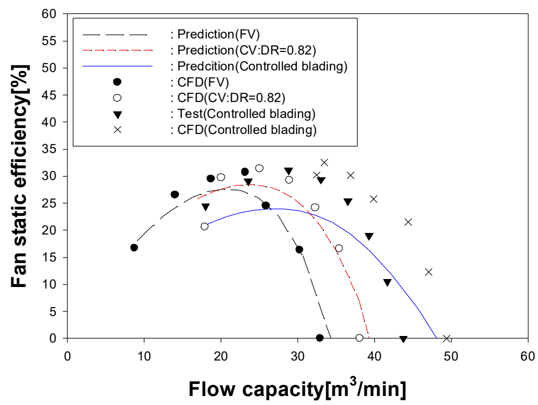



Fig. 4 Efficiency curves of three fan models

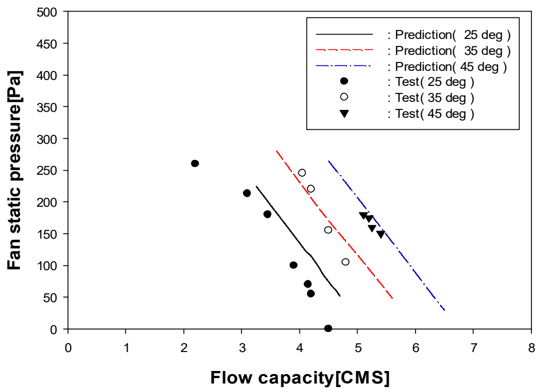



Fig. 5 Static pressure curves of variable-pitch fan

Fig. 5. shows the comparison between the FANDAS prediction and the measurement of a variable-pitch air-conditioning fan as second verification example( refer to Fig. 6 ). This fan is designed by the CBD method with the tip diameter of 0.63 m, the hub to tip ratio of 0.23, 1440 RPM and F-series airfoil thickness distribution. As shwon in Fig. 5, the FANDAS code predcition is favorably compared with the measurement of pressure curves of fan under variable-pitch operation. The results of Fig. 5 imply that the FANDAS code is also suitable for predicting fan performance at variable-pitch operation.

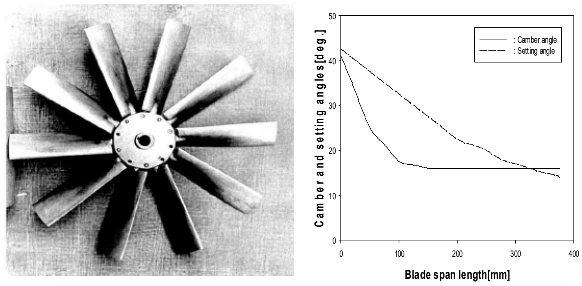



Fig. 6 Air-conditioning fan and its blade angles

## 3. VARIABLE-PITCH FAN DESIGNS BY DIFFERENT BLADING DESIGN METHODS

In order to examine and compare the effects of three different fan blading design methods, axial flow type jet fans are newly designed by applying the FANDAS code with the same design requrements and parameters. Three new fan models by the FVD, the CVD and the CBD methods are also compared with an exsiting model of the Samwon E&B of Fig. 7, which is designed by reverse-engineering method.

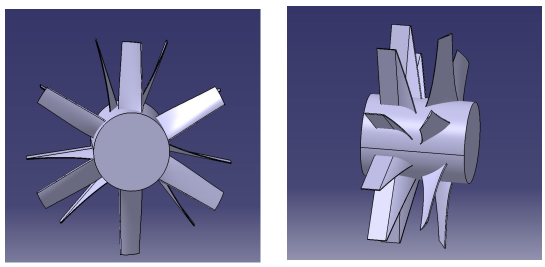


Fig. 7 Geometry of existing fan model

The variations of the setting and camber angles of the designed fans along blade span are shown in Fig. 8. It can be seen that the setting and camber angles of the CVD are somewhat gently compared to the FVD and the CBD results.

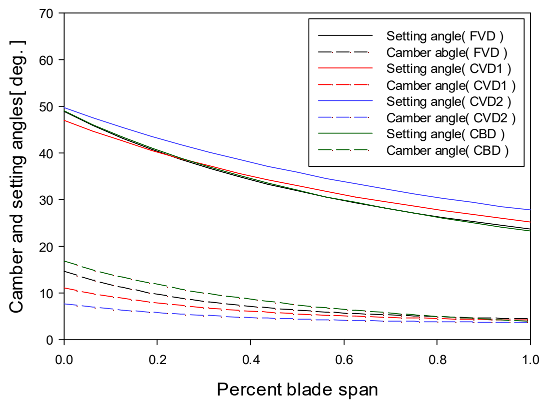



Fig. 8 Setting and camber angles of jet fan models

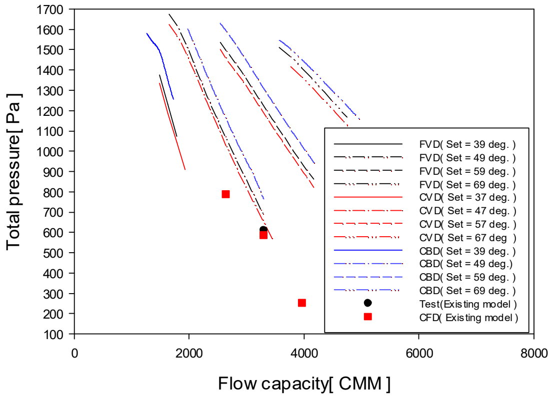



Fig. 9 Pressure curves of three jet fan models

Figs. 9 and 10 show the changes in total pressure and total efficiency according to the flow rate of the three jet fan models. As shown in Fig. 8, as the design hub setting angle (47, 49 degrees) of the rotor blade is reduced and increased, the flow rate range of the fan is significantly reduced. When the setting angle is reduced compared to the design value, the slope of the total pressure curve rapidly increases and the flow rate range is significantly reduced. Fig. 10 shows that the total efficiency decreases when the fan is used outside the design setting angle. It is noted that the FANDAS efficiency prediction is based on pure thermodynamic calculation, so tends to be somewhat higher than the actual value based on shaft power.

In addition, comparing the results of the three fan blade design methods, it can be seen that the CVD and the CBD methods are advantageous in terms of operating range, while the FVD and the CBD method are advantageous in terms of efficiency. As shown in Figs. 9 and 10, three fan models desined by the FVD, the CVD and the CBD methods shows superior performance and efficiency to the existing one.

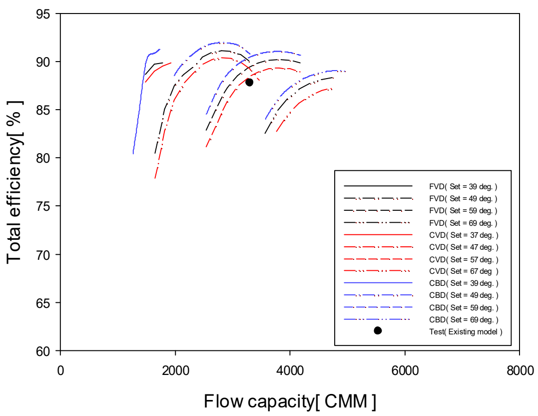



Fig. 10 Efficiency curves of three jet fan models

To sum up the whole results of Figs. 8, 9 and 10, it can be concluded that the CBD is more appropriate method than the FVD and the CVD in the aspects of fan operation range and efficiency.

## 4. OPTIMAL DESIGN OF A VARIABLE-PITCH FAN BY USING THE CBD METHOD

Based on the CBD method of the FANDAS code and the design result of section 3 of this paper as initial solution, a design optimization problem is constructed and solved with optimization algorithm of the PIAnO code[10] for maximum total efficiency of fan. In this optimal design problem, total 19 design variables including 10 blade section camber and setting angles are selected and 11 design constraints are used (refer to Fig. 11). After several iterations, all the design variables and constrains are converged and optimal design solution is achieved. Fig. 12 shows the 3-D geometries of rotor and stator blades of optimal fan.


Fig. 11 Optimization design scheme of axial fan

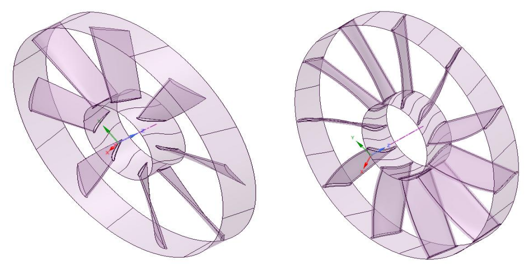
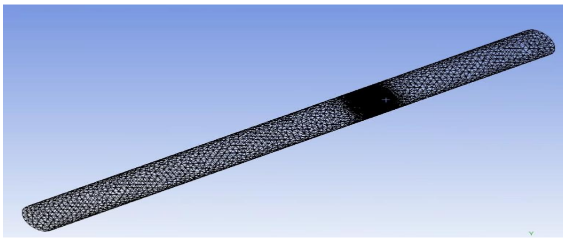


Fig. 12 Optimal variable-pitch fan design
( left: rotor blade, right: stator blade )

In order to verify the optimal fan design, CFD modeling is made with the unstructured mesh system of duct type flow domain of optimal fan shown in Fig. 13 and numerial computations are carried out by the ANSYS CFX code with frozen rotor scheme and k-ω SST turbulence models.



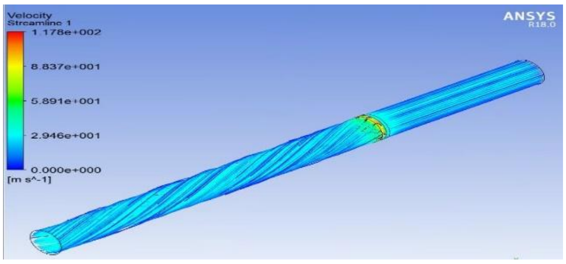



Fig. 13 Mesh system and streamlines of optimal fan model

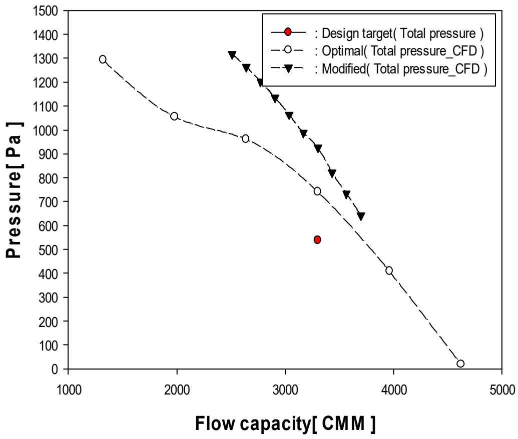



Fig. 14 Total pressure curves of fan models

Fig. 14 shows the predicted total pressure curves of optimal and modified fan models. Modified model is a derivative fan which can be easily constructed through some modification of setting angles of designed blade element sections. Because the CBD method determines and combines several blade section designs to form 3-D fan blade geometry, it is very flexible to design another new fan model by the setting combination of blade sections. As shown in Fig. 14, optimal fan model shows much higher total pressure at design point than the design target of 540 Pa and wide operation range from 1200 to 4600 CMM. Modified fan model also shows remarkable total pressure improvement compared with the optimal one through some setting angle modification.

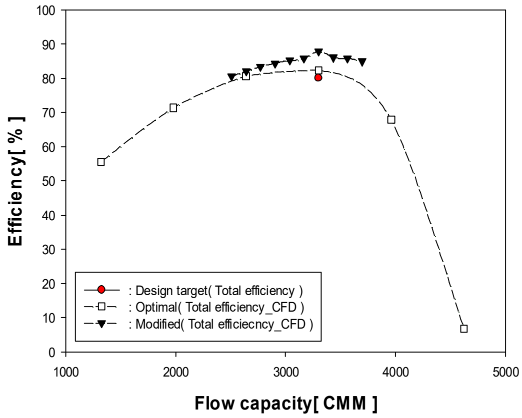



Fig. 15 Total efficiency curves of fan models

Total efficiency curves of optimal and modified fan models are predicted and compared in Fig. 15. As shown in Fig. 15. The efficiency predicted by CFD technique is based on shaft power. The design-point efficiency of optimal fan model is around 83% and higher than the design target of 80%, and the efficiency of modified fan model is much higher the design target of 80%. Total efficiencies of optimal and modified fan models are also maintained above 80% in wide range of flow capacity. Predicted fan power curves are plotted and compared in Fig. 16. The power of optimal fan is much lower than the design target of 55 kW at design point and maintained as lower level than 55 kW over entire flow capacity range.

From the CFD results of Figs. 14, 15 and 16, fan design optimization by the FANDAS code with the CBD method gives high performance and efficiency fan model which can operate in wide operation range and may be sutiable for variable-pitch operation.

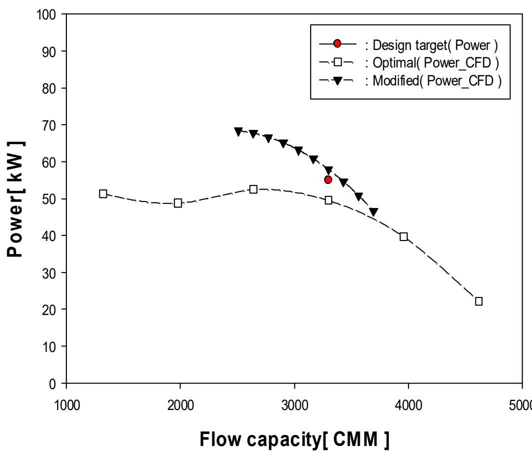



Fig. 16 Power curves of fan models

## 5. CONCLUSIONS

The present study examines the effects of three different fan blading design methods (the FVD, the CVD and the CBD) on fan performance, efficiency and operation range. From the design study results, the CBD is shown to be superior to the FVD and the CVD in the design of variable-pitch fan with wide operation range and high efficiency. Based on the CBD method, optimal design of a variable-pitch fan is conducted for maximizing fan total efficiency with multiple design variables and constraints. Optimal fan model shows larger total pressure, higher total efficiency and lower power than the design target values, and can be easily modified for obtaining another new fan model because of the design flexibility of the CBD method.


## 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)


## REFERENCES


[1] Wright, T., and Gerhart, P. M., 2009, *Fluid Machinery: Application, Selection and Design*, 2nd edition, CRC Press.

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

[3] van der Spuy, S. J., 1997, "Design of a Low-Noise Rotor-Only Axial Flow Fans," *Master Thesis of Mechanical Engineering, University of Stellanbsch.*

[4] van der Spuy, S. J. and von Backstrom, T.W., 2002, "Performance of rotor-only axial fans designed for minimum exit kinetic energy," *R & D Journal, vol. 18, no.3, pp.63-69*

[5] Lee, C., 2021,"A Performance Prediction Method of the Axial Flow Fans with Blade Sweep," *KSFM Journal, vol. 24, no.5, pp.24-29.*

[6] Dixon, S. L., 1998, *Fluid Mechanics and Thermodynamics of Turbomachinery,* 4th edition, Buterworth & Heinemann.

[7] McKenzie, A. B., 1997, *Axial Flow Fans and Compressors*, Ashgate.

[8] Lee, C., 2016, *Development of an integrated computer program for designing, performance, and noise analysis of axial flow fans for automobile,* Technical report, Hanon systems.

[9] ANSYS Inc., 2011*, ANSYS CFX manual*, ANSYS Release 16.0

[10] PIDOTEC, 2022, *PIAnO manual*