Conference Paper

Computer Axial Flow Fan Optimization Design

Hsin-Hung Lin[1] and Jui-Hung Cheng[2]

[1]Department of Creative Product Design, Asia University, Taichung, Taiwan
[2]Department of Mold and Die Engineering, National Kaohsiung University of Science and Technology, Kaohsiung, Taiwan

Abstract

In this study, the method of fuzzy multi-criteria decision-making is utilized for carrying out the assessment of axial-flow fan designs on the basis of competitive selection. Furthermore, the flow-field analysis by FLUENT has been conducted for comparison with the testing results by wind-tunnel measurements and experiments. With the modifications of geometric parameters, the internal three-dimensional flow fields of axial fans have been analyzed via the analysis by numerical simulation and a performance testing was conducted. The performance curves obtained have been compared with numerical results. And by taking the design optimization of axial-flow fans as a case, the validity of this design approach has been explained and verified. The research results revealed that this evaluation method not only incorporates both the practicability and the objectivity, it can also assist decision-makers in carrying out a strategic decision in complex and uncertain environments. Through the results of flow-field analysis, designers can effectively master the trend of flow-field distribution for axial-flow fans.

Keywords: method, fan design, numerical simulation, fan experiments

Corresponding Author:
Hsin-Hung Lin
hhlin@asia.edu.tw

Received: 29 August 2018
Accepted: 18 September 2018
Published: 11 November 2018

Publishing services provided by Knowledge E

Selection and Peer-review under the responsibility of the ICOI-2018 Conference Committee.

1. Introduction

Nowadays product requirements by consumers are more and more changeable, and no single product can satisfy the preference of consumers. To cater to consumer demands, the designers must subjectively determine the most relevant conceptual design idea under limited time and resources. However, during the assessment of design concepts, any subjective perception will affect the selection and therefore the designers may fail to make the most objective decision.

To resolve this, the authors have constructed an assessment framework of product designs. And the fuzzy synthetic evaluation (FSE) method has been conducted for the evaluation of product designs. To reduce the uncertainty and vagueness during the selection of designs, the simulation software FLUENT has been used for carrying

How to cite this article: Hsin-Hung Lin and Jui-Hung Cheng, (2018), "Computer Axial Flow Fan Optimization Design" in *The 2018 International Conference of Organizational Innovation*, KnE Social Sciences, pages 1630–1639. DOI 10.18502/kss.v3i10.3498

out the flow-field analysis (Ho,2009). In addition, to further understand the trend of flow-field distribution of axial fans (Hermes,2012), performance curves were acquired by wind-tunnel testing according to AMCA 210-85 (Lin,2004). The results have been compared with numerical ones to check the validity of this approach. The time for trial and error during product design and development is expected to be reduced by this approach and an objective and accurate result is obtained (Tsai,2007). A ranking of design schemes has been generated based on the design characteristics, and thus the ideal design can be determined.

2. Governing Equations

In three-dimensional Cartesian coordinates, the flow field can be solved by governing equations of continuity, momentum, and energy. Based on fundamentals of the finite-volume method, the computational domain must be partitioned into many small control volumes. After a volume integral, the equations of mass, energy, and momentum of fluids can then be transformed into algebraic equations for numerical calculations.

2.1. Theory of turbulence model

Since turbulence causes the exchange of momentum, energy, and concentration between fluids, this causes fluctuations. Such fluctuations are of small scale and with a high frequency. Therefore, when simulating turbulent flows (Choi, 2012), manipulations on the control equations are required for filtering out turbulence components which are at extremely high frequency or of extremely small scale (Owen, 2013). However the modified equations may comprise variables which are unknown to us, while the turbulence model requires known variables to confirm these variables (Shih, 2008). The standard $\mathbf{k}-\boldsymbol{\varepsilon}$turbulence model is selected for the calculation of flow fields in this study (Kennedy, 2013; Hurault, 2012).

2.2. Performance testing equipment for wind turbines

The main device of the performance testing equipment for fans is an outlet-chamber wind tunnel which conforms to AMCA 210-99. The principal parts include flow setting means, multiple nuzzles, flow-rate regulating devices(Chang,2010), etc. The major function is to supply a good and stable flow field for measurement, and to acquire the complete performance curves(Lin,2012;Greenblatt,2012).

2.3. Calculation of flow rates

With the pressure difference between nozzle outlet and inlet (and) being measured, the flow rates on cross-sections of nozzles (shown in Fig. 1) can be obtained by the nozzle coefficients. For the calculation of the outlet flow rate of the fan under test, the effect of density variations must be considered (Hurault,2012).

The equation for the calculation of flow rates in a test chamber with multiple nuzzles is

$$Q_5 = 265.7Y\sqrt{\Delta P/\rho_5}\sum_n \left(C_n A_{6n}\right) \tag{1}$$

Where Q_5 = the total flow rate measured by a bank of nozzles, CMM;

ΔP = the pressure difference across the nozzles, mm-Aq;

ρ_5 = the air density upstream of the nozzles, kg/m^3;

Y = expansion factor;

C_n = the discharge coefficient of the nth nozzle (Nozzle Discharge Coefficient);

A_{6n} = the cross-sectional area of the nth nozzle's throat, m^2.

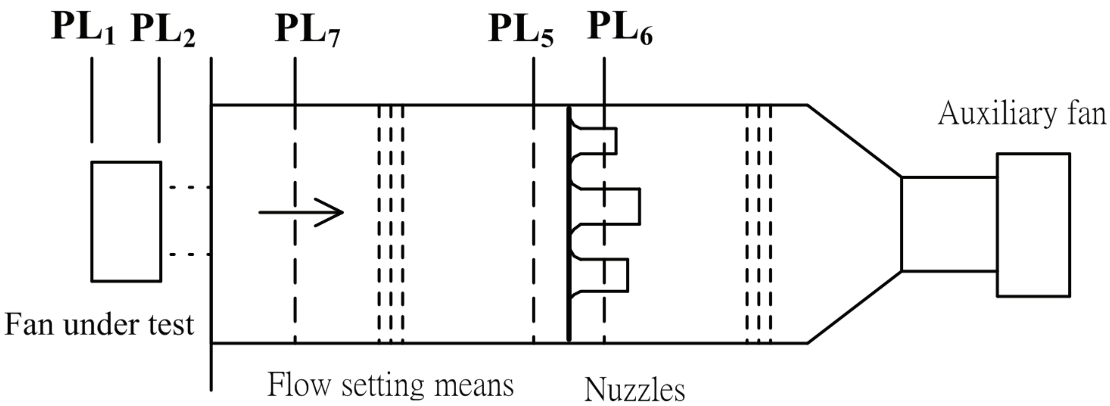


Figure 1: Schematic of measurement planes.

2.4. Calculation of air pressures

The static pressure happens to be equal to the static pressure obtained at the outlet test chamber P_{t_7}. This is only a special condition of said outlet test chamber when conducting the method of Type A (a test method with no duct at either the outlet or the inlet). When using different test methods or apparatus, the equation of the static pressure is thus more complicated. The equation of the dynamic pressure is:

$$P_{v_2} = \frac{\rho_2 V_2^2}{19.6} \tag{2}$$

Where P_{v_2} is the outlet dynamic pressure, mm-Aq;

V_2 is the outlet air velocity, m/s;

ρ_2 is the outlet air density, kg/m^3;

and

$$V_2 = \frac{Q_2}{60A_2} = \frac{Q}{60A_2} \cdot \frac{\rho}{\rho_2} = \frac{Q}{50\rho_2 A_2}$$

,

where Q_2 is the outlet flow rate, CMM;

Q is the standard flow rate, CMM;

A_2 is the outlet cross-sectional area, m^2;

ρ is the density of air at STP (1.2 kg/m^3).

$$P_t = P_s + P_v = P_s + P_{v_2}$$

$$P_t = P_s + \frac{\rho_2 V_2^2}{19.6} \quad (3)$$

2.5. Method of measuring the performance curves of fans

With a fixed amount of power, the flow rate varies inversely proportional to the output air pressure. Since the efficiency of fans changes as the flow rate varies, a non-linear relationship between the flow rate and the air pressure exists and this forms the performance curve of fans. The measurement process is shown in Fig. 2.

3. Case Verification and Analysis

3.1. Verification between numerical simulation and experiment testing

Mock-up samples were built for wind-tunnel testing and the results have been compared with those by simulation (as shown in Fig. 3). The geometric parameters of the final design are shown in Table 1.

It is known from Table 2 that the simulation results obtained are about 3% lower than the counterparts by experiment testing. After further investigation, it was found that this mostly comes from the difference between the real-test environment and the simulation configuration (as shown in Fig. 4). Since the resistance of a fan varies at different points of operation and the average rotation speed during the entire testing

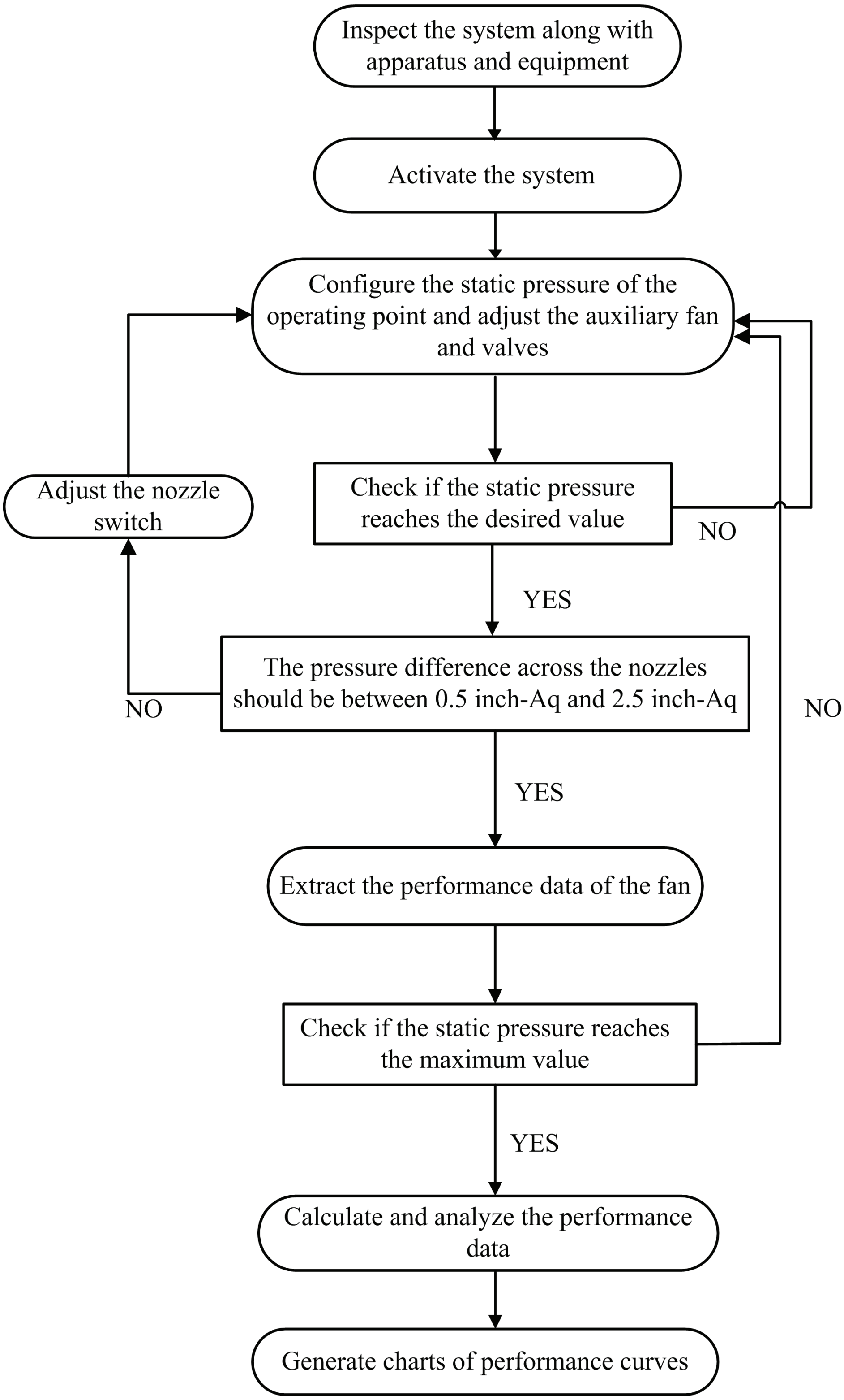


Figure 2: Operational flow chart of fan performance measurements.

process is 3% larger than design rotation speed, it is recommended increasing the rotation speed 2~4% and this can compensate for the tolerance.

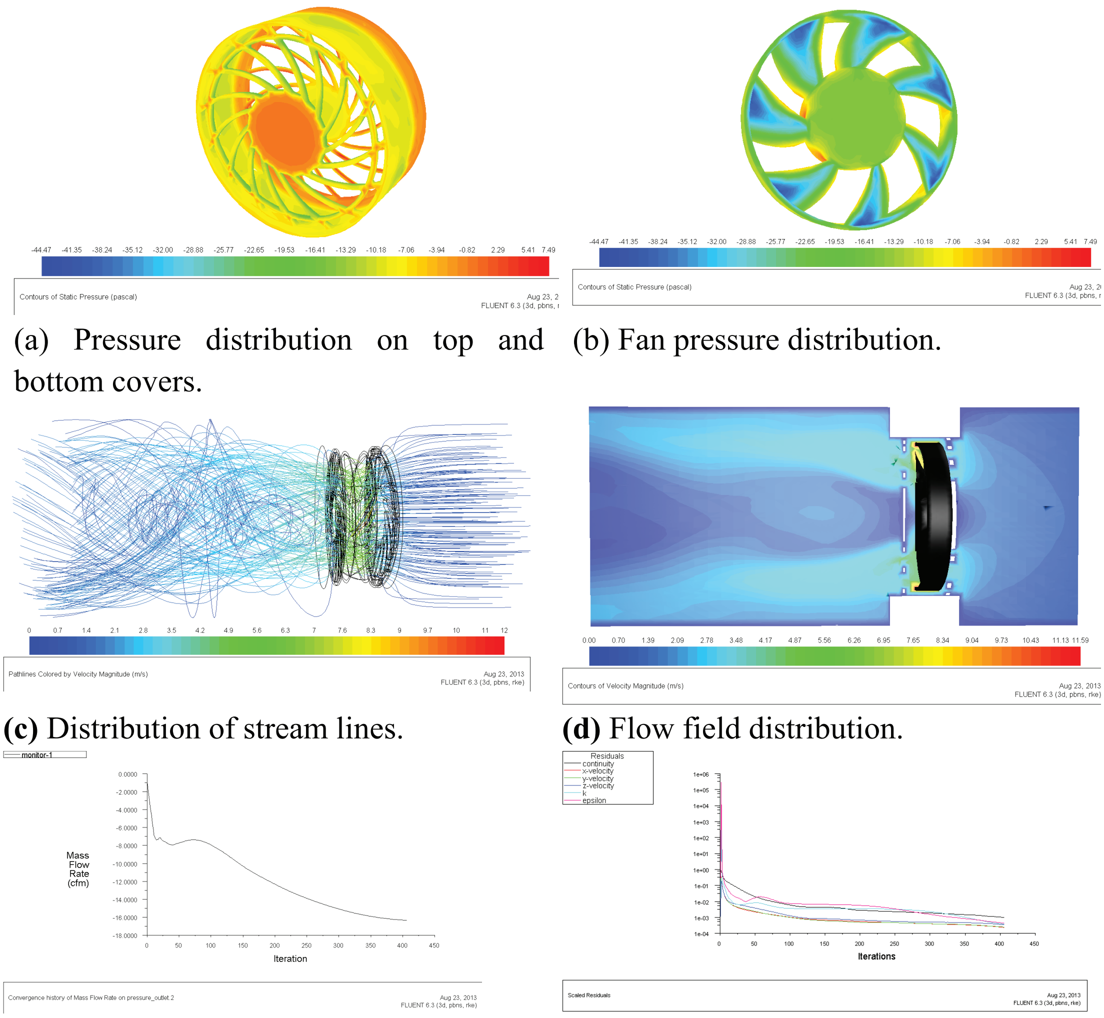
Table 1: Blade parameters.

Airfoil profile	NACA 65-Series
Airfoil name	NACA 65-Parabolic arc
Blade count	7
Hub radius	8
Blade-tip radius	36
Outside radius of housing	40
Blade tip clearance	0.75
Incidence angle of the blade at the hub	50
Incidence angle of the blade at the tip	35
Blade width at the hub	11
Hub thickness	23.5
Number of sections	31

Figure 3: Solid geometry.

3.2. Design cases and comparison between simulation results

The observation from a plane above the impeller intake indicates that the stream line distribution of NO.3 at the inlet is greatly affected by the turbulent flow. On the contrary,



(a) Pressure distribution on top and bottom covers.

(b) Fan pressure distribution.

(c) Distribution of stream lines.

(d) Flow field distribution.

(e) Number of iterations.

(f) Convergence plot.

Figure 4: Results of numerical simulation.

Table 2: Comparison between results by numerical simulation and experiment testing.

	Numerical simulation	Experiment testing	Tolerance
Flow rate	16.8CFM	16.3CFM	3%
Static pressure	1.75mm-H2O	1.71mm-H2O	2%

NO.4 is hardly affected and the air flows through the honeycomb structure and finally into the atmosphere. The flow field variations around the impellers of NO.1 and NO.2 are both very large. And recirculation occurs at the downstream for NO.1 and NO.3. This obviously impacts on the performance and results in lower flow rates Fig. 5.

It can also be discovered at the downstream prior to the outlet that, flows collide with each other behind the impeller region in NO.1 and NO.3 and this greatly affects the flow smoothness at the outlet. For NO.1, NO.2, and NO.3, the adverse influence of

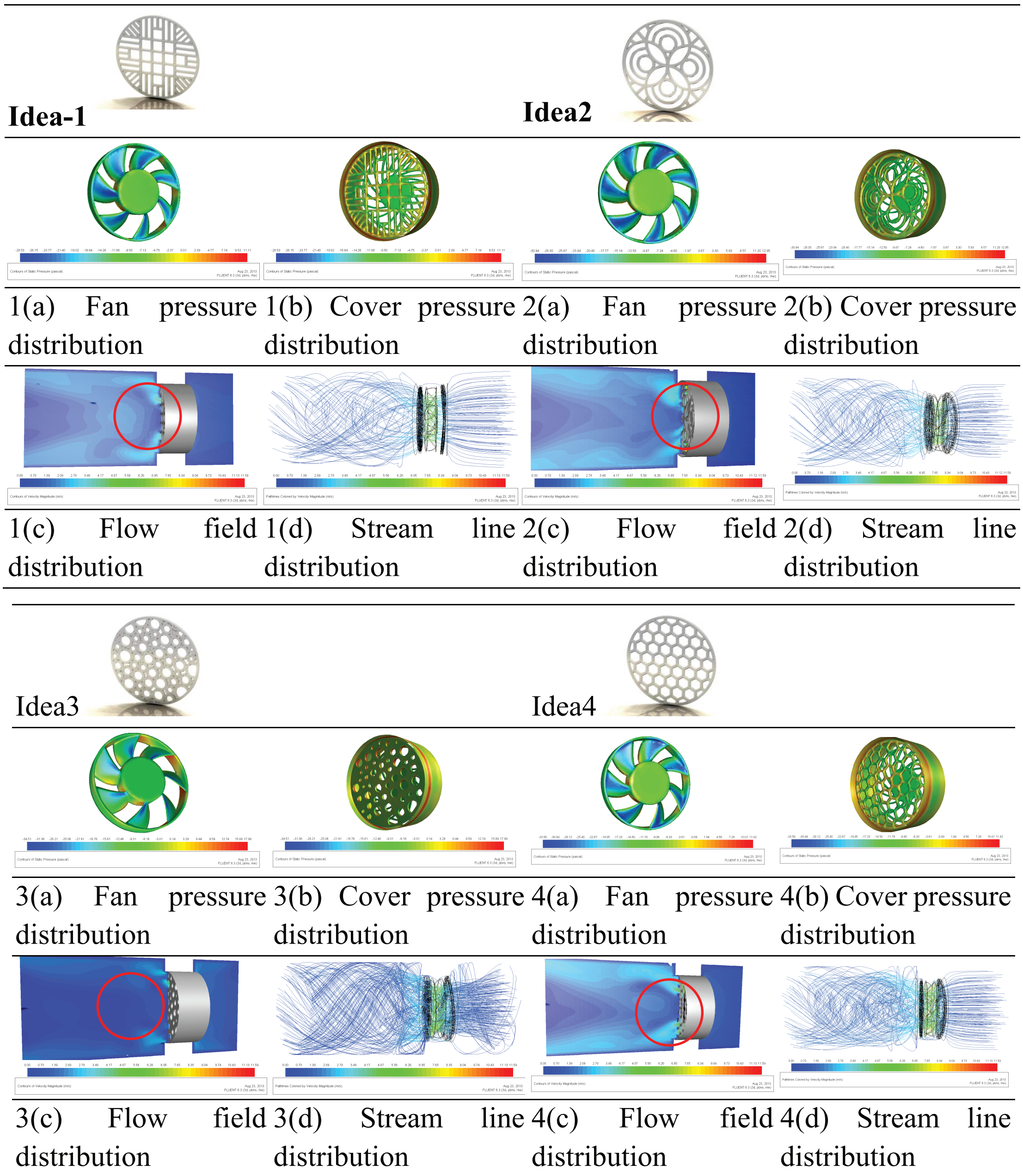


Figure 5: Simulation results of 4 design cases.

the external forms may contribute to the reduction of outlet flow rate. More than that, the vortices occur in NO.1, NO.2, and NO.3 disturb the smoothness of flow fields, and then affect the flow conditions downstream the impeller. The maximum flow velocity of NO.4 is higher than those of NO.1, NO.2, and NO.3. This also indicates that the flow field of NO. 4 is smoother and the flow rate obtained is also the highest. After the fluid flows through the blades, it may subject to the influence of the outlet profiles. The geometry of NO.4 creates less resistance and facilitates the airflow all the way to the outlet. A higher flow rate is the evidence of the smoothness of the airflow.

4. Conclusions and Suggestions

In this study, numerical analysis was performed to obtain the performance curve of the design model, and the results were further compared with the actual measurement results. Due to the maximum weight obtained, the NO.4 model proved to be the best design in all design options. The engineering evaluation needed for the overall design of any fan product can be focused on this approach by focusing on the overall overview of the design solution and new methods of evaluating performance. In other words, in addition to the improvement of the fan profile, the entire evaluation method has been improved, which involves related complexity. For further research, it is recommended to emphasize fan performance based on cost considerations. When redesigning fan products, the concept and structure of any new design can be studied through the evaluation methods proposed in this study, and the time and effort spent on actual testing can be reduced.

References

[1] Chang,C.C.Kuo, Y.F.Wang, J.C. andChen, S.L.(2010). Air cooling for a large-scale motor, Applied Thermal Engineering (30) 1360-1368.

[2] Choi,J.Jeong,Yoo, M. J. andSeo, M.(2012). A new CPU cooler design based on an active cooling heatsink combined with heat pipes, Applied Thermal Engineering (44) 50-56.

[3] Greenblatt, D.Avraham, T. andGolan, M.(2012). Computer fan performance enhancement via acoustic perturbations, International Journal of Heat and Fluid Flow(34)28–35.

[4] Hermes,J.L.Jr, L.S andCastro, A.G.(2012). Thermal-hydraulic design of fan-supplied tube-fin condensers for refrigeration cassettes aimed at minimum entropy generation, Applied Thermal Engineering (36) 307-313.

[5] Ho,S.H.Rosario, L. andRahman, M.M.(2009). Thermal comfort enhancement by using a ceiling fan, Applied Thermal Engineering (29) 1648–1656.

[6] Hurault,J.Kouidri, S. and Bakir, F.(2012). Experimental investigations on the wall pressure measurement on the blade of axial flow fans, Experimental Thermal and Fluid Science (40) 29-37.

[7] Hurault,J.Kouidri, S.Bakir, F. andRey, R.(2012). Experimental and numerical study of the sweep effect on three-dimensional flow downstream of axial flow fans, Flow Measurement and Instrumentation (21)155–165.

[8] Kennedy,I. J.Spence, W.T.Spratt, G.R. andEarly, J.M.(2013). Investigation of heat exchanger inclination in forced-draught air-cooled heat exchangers, Applied Thermal Engineering (54) 413-421.

[9] Lin,S.C. andChou, C.A.(2004). Blockage effect of axial-flow fans applied on heat sink assembly, Applied Thermal Engineering (24) 2375–2389.

[10] Lin,S.C. andTsai, M.L.(2012). An integrated performance analysis for a backward-inclined centrifugal fan, Computers and Fluids (56) 24-38.

[11] Owen,M. andKröger, D.G.(2013). Contributors to increased fan inlet temperature at an air-cooled steam condenser, Applied Thermal Engineering (50) 1149-1156.

[12] Shih,Y.C. Hou, H.C. andChiang, H. (2008). On similitude of the cross flow fan in a split-type air-conditioner, Applied Thermal Engineering (28) 1853–1864.

[13] Tsai,B.J.andWu, C.L.(2007). Investigation of a miniature centrifugal fan, Applied Thermal Engineering (27) 229-239.