International Journal of Air-Conditioning and Refrigeration Vol. 17, No. 2 (2009) 68~73

Optimum Design of Volute Configuration in a Sirocco Fan using CFD and DOE

Uk-Hee Jung, Young-Seok Choi[†*], Kyoung-Yong Lee[*]

Tea Sung ANST, Inc., Seoul 152-847, Korea
[*]Thermal & Fluid System Team, Korea Institute of Industrial Technology, Cheonan 330-825, Korea

(Received January 12, 2009; Revision received May 15, 2009; Accepted June 16, 2009)

Abstract

In this paper, a numerical study has been carried out to investigate the influence of volute geometries on the performance of a sirocco fan. In order to achieve an optimum volute design and explain the interactions between the different geometric configurations in the volute system, three-dimensional computational fluid dynamics and the 'design of experiment' method have been applied. Several geometric parameters, such as the volute expansion angle, the cut-off position and the bell mouth shape, are employed to improve efficiency and performance. 2^k factorial designs were performed to screen the most influential parameters and interactions, and showed that the cut-off position and the bell mouth shape are the most significant parameters. The optimum design was selected as a result of the response surface methodology, and effects of these parameters and their interactions were presented. From the results of computational analyses and experimental data, the performance and efficiency of the sirocco fan were successfully improved. Also, detailed effects of geometric variables of the volute system on the fan performance were discussed.

Key words: Optimum design, Volute, Sirocco fan, CFD (Computational Fluid Dynamics), DOE (Design of Experiment)

Nomenclature

A : Starting angel of cut-off [deg]
D : Impeller diameter [mm]
D_1 : Bell mouth diameter [mm]
Dc : Distance between the cut-off and the exterior diameter of impeller [mm]
N : Fan speed [rev/s]
P : The power drawn by the fan [w]
Q : Volumetric flow rate [m^3/s]
R : Distance between the scroll of volute and the center of impeller [mm]
R_0 : Distance between cut-off and the center of impeller [mm]
a : Scroll expansion angle [deg]
Δp : Pressure rise across the fan [Pa]
θ : Sweep angel of the scroll [deg]
Φ : Flow coefficient
Ψ : Pressure coefficient
ξ : Power coefficient
η : Efficiency
ρ : Air density [kg/m^3]

Superscript

k : Number of project variables

1. Introduction

The design of a centrifugal fan is traditionally based on the classical methods of Stepanoff(1) and Eck(2). Recently, through use of improved facilities are available for performing numerical calculations, more researchers have become interested in considering viscous flow modeling by solving Navier-Stokes equations. CFD tools can help us understand the flow-field variations throughout a machine and are also very efficient for the study of the effects of geometrical parameters on the performance of fans.

Fig. 1 shows the flowchart of this research. Initially, we established a reference sirocco fan model, and we choese geometric parameters which are going to

---

[†] Corresponding author. Tel.: +82 41 589 8337, Fax.: +82 41 589 8330
E-mail address: yschoi@kitech.re.kr

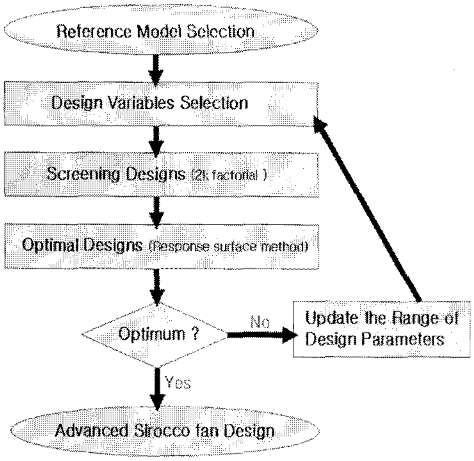

Fig. 1. Optimization process flowchart.

affect the performance of a sirocco fan. Next we analyzed the influence of geometric parameters using the design of experiment (DOE) method, and synthesized and produced the most suitable volute model. Finally, the optimum sirocco fan model and the reference sirocco fan model were compared in performance tendency through experiment results and numerical analysis results.

In this study, the DOE using CFD is applied to the design of a sirocco fan to maximize its performance. To reduce computing time, we omitted the repeat and block methods in DOE Fig. 2 Geometric parameters of a sirocco fan because there was no repeat error of an experiment in CFD.

2. Reference model description

The geometrical shape of the volute in the sirocco fan is shown in Fig. 2. We created several configurations of the diffuser with different scroll expansion angles to reduce flow loss. The following exponential function is used in the scroll shape design.

$$R = R_0 \exp(\theta \times \tan a) \tag{1}$$

The CFD solves Reynolds averaged Navier -Stokes equations, using turbulence models to compute the averaged turbulent stresses. One among the most effective turbulence models is the shear stress transport (SST) turbulence model$^{(3)}$. The model works by solving a k- ω model at the wall and k-ε model in the bulk flow. A blending function ensures a smooth transition between the two models. The SST turbulence model more accurately predicts the separation zone than other models, and combines the advantages of both the standard k-ε model and the k-ω model. Therefore we selected the SST turbulence model for this research to get an accurate performance prediction of the sirocco fan.

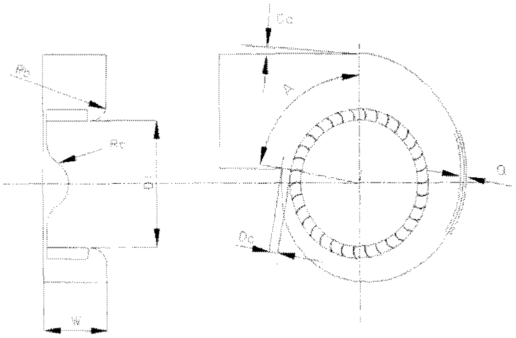

Fig. 2. Geometric parameters of a sirocco fan.

3. Design of experiments (DOE)

Design of experiments$^{(4)}$ is a structured, organized method that is used to determine the relationship between the different design variables and the responses. When the results of these experiments are analyzed, they help to identify optimal conditions of project variables as well as details such as the existence of interactions and synergies between project variables.

3.1 Screening designs

Factorial designs in screening designs allow for the simultaneous study of the effects that several project variables may have on a process. When performing an experiment, varying the levels of the project variables simultaneously rather than one at a time is efficient in terms of time and cost, and also allows for the study of interactions between the project variables.

To minimize time and cost, we used fractional factorial designs with four project variables that exclude some of the factor level combinations. Some of the effects will be confounded, and the confounded effects cannot be estimated separately. We selected four project variables (A, *a*, D1 and Dc) for fractional factorial designs, and other parameters are fixed. The variation ranges of each design parameter is limited to about 5~10% at center points of the reference model. For an efficient analysis, we used the commercial statistics analysis program, Minitab. Table 1 shows experimental sets where 2^k factorial designs are applied. Project responses in 2^k factorial designs are total pressure and efficiency at the outlet of a sirocco fan, and the results of CFD are total pressure and efficiencies are in Table 1.

To analyze project responses in 2^k factorial designs, we used main effects plots in Minitab. Main effects plots are shown in Fig. 3 and Fig. 4. The main effect in this study occurs when the mean response changes across the levels of a geometric factor. We used main effects plots to compare the relative project responses (total pressure and efficiency) of the effects across the factor. As a result, the effect of α and Dc are much smaller than the effect of A and Dc. Also, project responses remain virtually the same when the Dc and α are moved from a low level to a high level and increase when A is moved from high level to low level. So, we carefully selected A and D1 for the parameters of the response surface method because these parameters are the main influences of project responses.

Table 1. Numerical analysis sets of 2^k factorial designs.

2^k No	D1 (mm)	A (deg)	Dc (mm)	α (deg)	Pt (Pa)	eff (%)
1	190	45	14.88	5	2927.0	66.66
2	198	45	14.88	7	2944.9	66.62
3	190	65	14.88	7	2768.0	59.06
4	198	65	14.88	5	2805.5	58.05
5	190	45	19.84	7	2898.1	65.70
6	198	45	19.84	5	2964.5	68.33
7	190	65	19.84	5	2796.9	58.91
8	198	65	19.84	7	2862.0	62.04

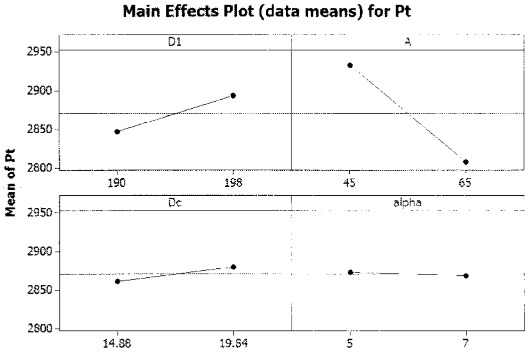

Fig. 3. Main effects plot for total pressure.

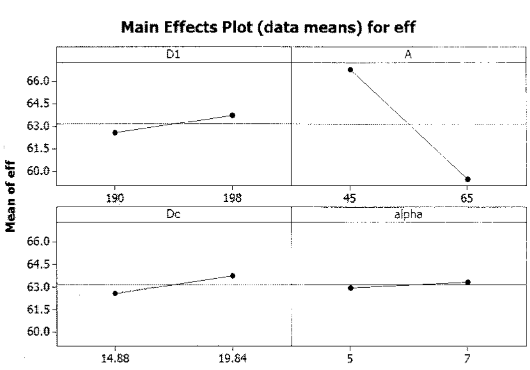

Fig. 4. Main effects plot for efficiency.

3.2 Response surface methodology (RSM)

RSM⑸ explores the relationships between several explanatory variables and one or more response variables. A central composite design (CCD) in RSM methods can be implemented to estimate a second-degree polynomial model, which is still only an approximation at best. However, the second-degree model can be used to optimize a project response.

The CCD with two project variables for this study are shown in Fig. 5. The CCD of this study consists of four points in the factorial portion, four points in the axial portion and one center point of the design. Four additional points be added for a correct estimate. Consequently, 13 experiment sets were created for numerical analysis of RSM. Table 2 shows experimental sets where RSM are applied. Project variable values in numerical analysis sets of RSM are total pressure and efficiency at outlet of a fan, and the results of CFD presented as total pressures and efficiencies in Table 2.

To find the optimum range of the volute shape, we used an overlaid contour plot. Overlaid contour plots show how response variables relate to two continuous design variables while holding the rest of the variables in a model at certain settings. Fig. 6 shows an overlaid contour plot for two responses, total pressure and efficiency. The white region on the plot is the feasible region, or the area that satisfies the criteria for

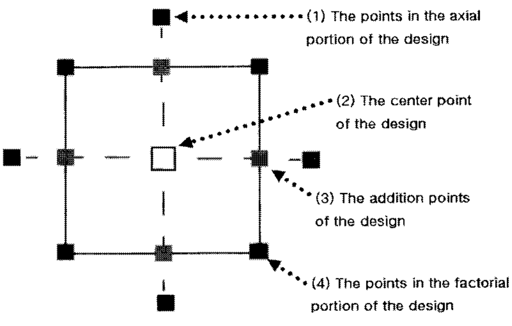

Fig. 5. The CCD with two project variables.

Table 2. Numerical analysis sets of RSM.

RSM No	D1 (mm)	A (deg)	Pt (Pa)	eff (%)
1	196.5	41.5	2954.1	68.153
2	203.5	41.5	2979.8	68.265
3	196.5	48.5	2934.9	66.631
4	203.5	48.5	2960.1	66.813
5	195.05	45.0	2939.8	67.331
6	204.95	45.0	2980.1	67.811
7	200.0	40.0503	3000.8	69.525
8	200.0	49.9497	2989.3	67.728
Center	200.0	45.0	2999.6	68.718
Add 1	196.5	45.0	2946.2	67.510
Add 2	200.0	41.5	2995.1	69.243
Add 3	200.0	48.5	2981.4	67.841
Add 4	203.5	45.0	2971.9	67.729

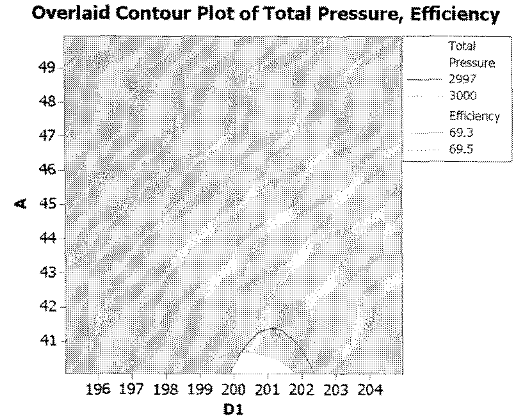

Fig. 6. Overlaid contour plot of total pressure and efficiency.

all responses. This means that if we set our project variables at any of the levels shown in the white area, the responses should fall within the specified ranges. Therefore we can confirm predicted project responses ranges where the total pressure is more than 2997 Pa, and efficiency is more than 69.3%.

3.3 Optimal designs

Response optimization is used to help identify the combination of input variable settings that jointly optimize a set of project responses. Joint optimization must satisfy the requirements for all the responses in the set, which is measured by the composite desirability. We calculated an optimal solution and drew a response optimization plot, Fig. 7. The plot shows the configuration of optimum volute design where D1 is 200 mm, A is 40.0503 deg. This set equals the RSM 7 model of numerical analysis sets, so we selected it as the optimum volute design of this study.

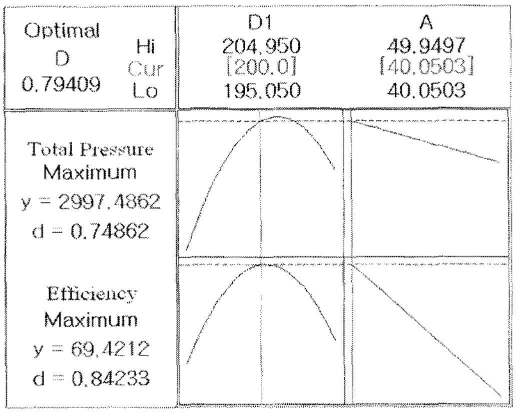

Fig. 7. Response optimization plot.

4. Experiment results

The fan under consideration is a forward- curved centrifugal fan with a conventional rotor design. The fan is tested in accordance with the laboratory method of testing fans$^{(6)}$. The pressure rise and flow rate are measured in the test chambers using conventional techniques. Performance data for the sirocco fan, which include pressure rise and shaft power are collected at a nominal operating speed (3550 rpm) and a range of flow rates. Due to minor variations in speed and air temperature, all data are corrected to the nominal rotational speed and standard atmospheric density.

CFD results are obtained for a range of flow rates so that fan performance data is generated and compared with the available experiment data. The following non-dimensional parameters$^{(7)}$ are used to characterize the fan performance :

$$\Phi = Q / ND^3 \quad (2)$$

$$\Psi = \Delta p / \rho N^2 D^2 \quad (3)$$

$$\xi = P / \rho N^3 D^5 \quad (4)$$

$$\eta = \Phi \Psi / \xi \quad (5)$$

The power is computed from CFD results by multiplying the torque on the fan blades by the angular speed of the impeller.

In Fig. 8, the pressure rise coefficient is plotted as a function of flow coefficient, and CFD results on the

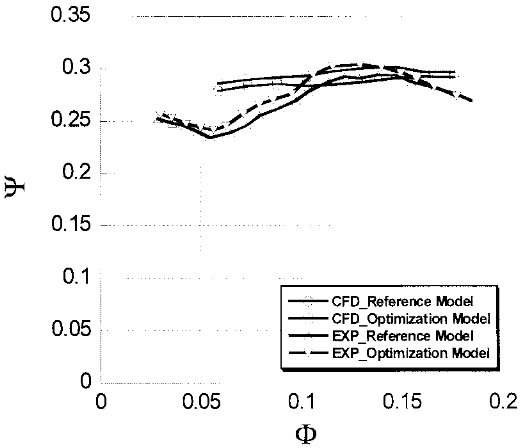

Fig. 8. Pressure coefficient vs. flow coefficient.

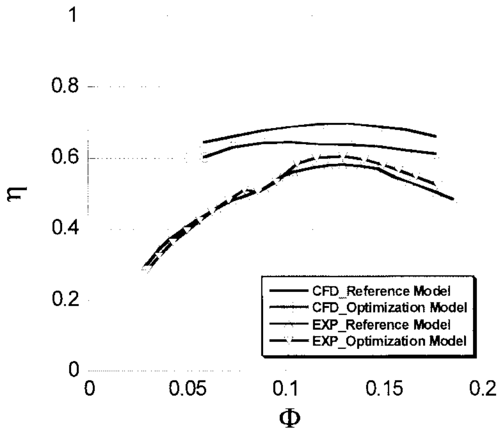

Fig. 9. Efficiency vs. flow coefficient.

optimized model using DOE methods are in agreement with experiment results at the design point (Φ is about 0.12). The flow in the fan is fairly steady and uniform at the design point, as well as at higher flow rates. At lower flow rates, however (not shown), the flow appears to be stalled in some of the rotor passages as a result of localized recirculation zones.

Fig. 9 shows the efficiency for the reference model and optimized model with the flow coefficient. The trend of efficiency variation along the flow coefficient were well predicted by the CFD. However, the overall levels of the predicted efficiency are higher than those of the experiment throughout the range of flow rates. The difference in the power coefficient results is reflected (because the efficiency of the motor is excepted from efficiency results); however, the point of peak efficiency is correctly predicted. Therefore, there is no problem in applying this to the design point of this research. While the present calculations are reasonable approximations of the mean flow field within the fan, it is to be expected that as the flow begins to break down at very low flow rates, the flow is highly unsteady. To compare performance rise in total pressure, CFD results show a 4.3% increase and experiment results show a 3.7% increase at the design point. Moreover, to compare performance rise in efficiency, CFD results show an 8.3% increase and experiment results show a 4.2% increase at the design point.

5. Conclusion

To summarize, the DOE method using CFD is applied to the optimum volute design of the sirocco fan. Performance calculations for the sirocco fan are carried out using a commercial CFD solver on an unstructured, hybrid mesh. The numerical results are found to be in good agreement with the available experimental results except in the region of low flow rate. Moreover, important performance trends, such as the change in pressure rise and efficiency versus flow rate and the point of peak efficiency, are correctly predicted. These results suggest that CFD is a useful tool in the design of a sirocco fan. Also, The DOE method helps us to improve our design processes, and understand how the geometric factors interact and drive the calculation process. The DOE using CFD is confirmed to be an efficient method for the optimum volute design of a sirocco fan.

References

[1] Stepanoff A. J., 1957, Centrifugal and Axial Flow Pumps; Theory, Design & Application, Wiley, New York.

[2] Eck, B., 1973, Fans Design and Operation of Centrifugal, Axial-Flow, and Cross-Flow Fans, Oxford, Pergamon Press, New York.

[3] Menter, F. R., 1993, Zonal Two Equation k-ω Turbulence Models for Aerodynamic Flows, AIAA Paper 93-2906.

[4] Hicks, C. R., Turner and Jr., K. V. 1999, Fundamental Concepts in the Design of Experiments, Fifth Edition, Oxford University Press, New York.

[5] Myers, R. H. and Montgomery, D. C. 2002, Response Surface Methodology : Second Edition, A Wiley- Interscience Publication, United States of America.

[6] ANSI/AMCA Standard 210-99, Laboratory Method

of Testing Fans for Aero dynamic Performance Rating, Air Movement and Control Association International Inc.

[7] Wright, T. 1999, Fluid Machinery : Performance, Analysis, and Design, CRC Press, Boca Raton London New York Washington, D.C.