Research Journal of Applied Sciences, Engineering and Technology 6(13): 2436-2442, 2013
DOI:10.19026/rjaset.6.3719
ISSN: 2040-7459; e-ISSN: 2040-7467

Submitted: December 20, 2012 Accepted: January 25, 2013 Published: August 05, 2013

Research Article

Numerical Optimization of Impeller for Backward-Curved Centrifugal Fan by Response Surface Methodology (RSM)

Fannian Meng, Quanlin Dong, Yan Wang, Pengfei Wang and Chunxi Zhang
School of Instrumentation Science and Opto-Electronics Engineering, Beihang University, Beijing, 100191, China

Abstract: A numerical optimum study on three-dimensional unsteady viscous flow in a centrifugal fan with backward-curved blades was performed. The influence of the inlet angle, the outlet blade angle and blade number on aerodynamic performance of the centrifugal fan was analyzed concerning the whole impeller-volute configuration. Response Surface Methodology (RSM) based on a three-level, three -variable Box-Behnken Design (BBD) was used to evaluate the interactive effects of factors such as inlet blade angle (37°-41°), outlet blade angle (61°-65°) and number of blade (10-14) on the efficiency. The optimum factors deriving via RSM were: inlet blade angle 37°, outlet blade angle 61.7° and blade number 14. The maximum efficiency was 93.7%, under optimum conditions. The method of reliable CFD technique in combination with RSM can be effectively applied to the improvement and experiment design of centrifugal fans.

Keywords: Centrifugal fan, CFD, impeller, numerical optimization, RSM

INTRODUCTION

As one of universal machines, large centrifugal fans have been broadly used in lots of departments of the national economy, such as cement stove ventilating system, mile ventilating system and so on. As important auxiliary equipments, fans consumed 30 percent of plant electrical consumption. So the study and optimization of centrifugal fan to improve the efficiency are important for the energy-saving of plant. Flow in a centrifugal fan is a complex three-dimensional phenomenon involving boundary layer, separation, secondary flow, turbulence, etc. In addition, the geometry is very complex. These geometrical parameters have significant influences on the efficiency of centrifugal fans.

A large number of researches were carried out to optimize the geometries of centrifugal fan. Bayomi *et al*. (2006) researched the effect of inlet straighteners on the performance characteristics of centrifugal fans. Two types of straighteners were used in the research, circular tubes and zigzag cross section, with different lengths. Circular tubes with different diameters have been investigated. The study was conducted on three types of fans, namely radial, backward with exit blade angles 60^{o} and 75^{o} and forward with 105^{o} and 120^{o}. The results confirm that the inlet straighteners exhibit different effects on the fan performance for the different blade angles (Bayomi *et al*., 2006). Son *et al*. (2012) researched the effects of inlet radius and bell mouth radius on flow rate and sound quality of centrifugal blower. In the research, a total of eight numerical models were prepared by combining different values of bell mouth radius and inlet radius (the cross section of bell mouth was chosen as a circular arc in this research). The inlet radius was then revealed to have significant impact on flow rate with the maximum difference between analyzed models was about 4.5% while the bell mouth radius had about 3% impact on flow rate (Son *et al*., 2012). Morinushi (1987) studied the effects of the width-inner diameter ratio of an impeller, axial clearance between fan inlet nozzle and impeller shroud plate, blade setting angle, blade pitch-chord ratio and the spiral extension index of the volute on a forward-curved centrifugal fan. He suggested 0.8 or less for the ratio of impeller width to inner diameter, an optimal blade setting angle equal to 26° and two optimal values for the pith-chord ratio (0.7 and 1.1) for the small flow region. He also found that the axial clearance does not significantly influence the noise level in the range of 3%-10% of the impeller width. However, the spiral extension index has an important effect on aerodynamic performance and noise characteristics (Morinushi, 1987). Sheam-Chyun and Chia-Lieh (2002) designed a high-performance cooling fan under the space limitations of notebook computers with the emphasis on the blade shape, blade inlet angle and the outlet geometry of the housing. Zhao *et al*.

Corresponding Author: Fannian Meng, School of Instrumentation Science and Opto-electronics Engineering, Beihang University, Beijing, 100191, China

(2005) analyzed the effects of blade inlet angle and impeller gap. He found that the blade inlet angle and the impeller gap play an important role for fan performance (Zhao *et al*., 2005). Kwang-Yong and Seoung-Jin (2006) presented the response surface optimization method using three-dimensional Navier-Stokes analysis to optimize the shape of a forward-curved blades centrifugal fan. Behzadmehr *et al*. (2006) studied the effects of the entrance parameters and their interactions on the noise reduction, static head, mass flow rate, rotational speed and efficiency of a backward-curved centrifugal fan. Design of Experiment (DOE) method has been performed to analyze the results and their effects. Three different levels for the entrance curvature and two levels for the shape of blades passage, shroud, number of blades and inlet bell position have been selected to design and construct different fans for the aero-acoustic experimental measurements. The results analysis shows that to have a lower noise and higher efficiency, higher entrance curvature, hyperbolic profile for the shape of shroud and inlet bell position 1 (no radial clearance) must be selected. Li *et al*. (2011) investigated the influence of enlarged impeller in unchanged volute on G4-73 type centrifugal fan performance. He found that the flow rate, total pressure rise, shaft power and sound pressure level have increased, while the efficiency have decreased when the fan operates with larger impeller (Li *et al*., 2011).

The above research focuses on finding the effect of geometries parameters on the flow rate of centrifugal blowers. But few papers did research on the impact of inlet blade angle, outlet blade angle and blade numbers on the performance of fan by numerical method. We all know that the blade affects the performance of the fans largely. In the blade design, inlet blade angle, outlet blade angle and blade number have a direct impact on the fan performance, but it is still difficult to calculate it by theoretical methods. Luckily, at present, most of the experimental works could be replaced by Computational Fluid Dynamics (CFD) simulation because of the significant progress of computing technology. Thus, a new design process, or CFD-based design process, has been created. Instead of making a real model for experiments, computer models were built by CAD software. CFD programs, such as CFX, are used to simulate the flow after the meshing process. Typical models were thereafter prepared to validate the CFD results. The geometries of the model could be easily modified so that a number of models could be simulated to obtain the optimum model without the huge accompanying expense of research.

In our research, the CFD software, CFX is used to simulate the impellers with different inlet blade angle (from 37° to 41°), outlet blade angle (from 61° to 65°) and blade numbers (from 10 to 14). But in the simulation, we can only choose the situation that one parameter changes when the other two parameters are fixed. In this case, its disadvantage is that it does not include the interactive effects among the variables studied. As a consequence, the simulation does not depict the complete effects of the parameter on the efficiency. As a solution, the statistical method of Response Surface Methodology (RSM) has been proposed to include the influences of individual factors as well as their interactive influences by fitting of a polynomial equation to the simulation data. RSM which is a technique used for designing simulation helps researchers to build models, evaluate the effects of several factors and achieve the optimum conditions for desirable responses in addition to reducing the number of simulation. In this study, the response surface method was applied to the aerodynamic design of a backward-curved blades centrifugal fan to maximize the flow efficiency. Three geometric variables, i.e., inlet blade angle, outlet blade angle and blade number and one operating variable, i.e., efficiency were selected as design variables.

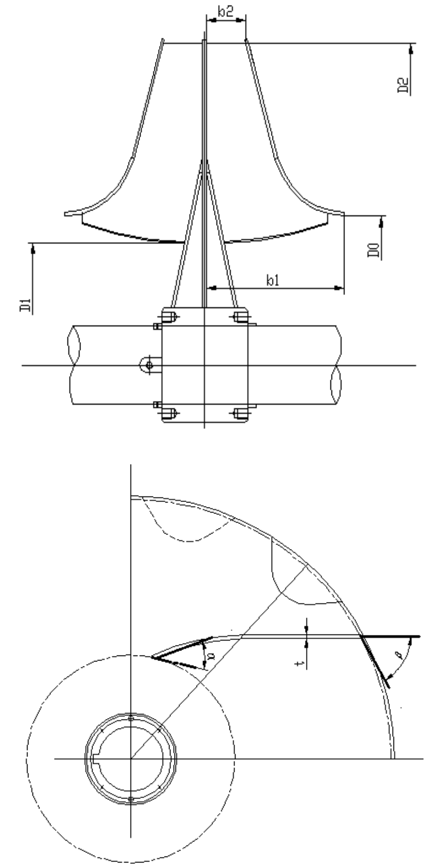


Fig. 1: Geometry of the backward-curved centrifugal fan

SIMULATION METHOD AND THEORY

Basic parameters of impeller: In this study, we utilized an industrial centrifugal fan (Fig. 1) as the research target. Table 1 shows the main dimensions of the impeller.

Table 1: Impeller dimensions

Geometirc structures	Dimensions
Impeller blade outlet diameter, D_2 (mm)	2880
Impeller blade inlet diameter, D_1 (mm)	1152
Impeller inlet diameter, D_0 (mm)	1334
Impeller outlet width, b_2 (mm)	187
Impeller inlet width, b_1 (mm)	682
Blade thickness, t (mm)	24

Table 2: The grid number

Item	Inlet region	Impeller	Volute
Number	127832	756849	308887

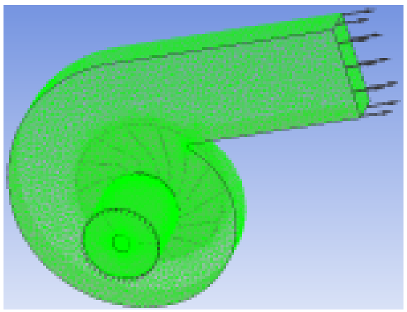


Fig. 2: Meshes of the centrifugal fan

Computational domain and computational grid: For this study, the three-dimensional fan models were first created in computer-aided design software (PROE) and exported into *.x-t files. The *.x-t files were then imported into ICEM, the mesh generator. In ICEM, model construction and split were processed. It consists of the inlet region, impeller and the volute. In order to simulate the real inlet condition, an extended cylinder is added to the inlet region for the purpose of imposing inlet boundary conditions. Grid independency tests were performed for each model discussed in this study. The static pressure which is equal to the atmospheric pressure was then given as the inlet boundary condition. The outlet boundary condition was set as mass flow rate. Unstructured tetrahedral cells are used for the meshing of impeller and volute, while hexahedron cells are used for the meshing of inlet region. The grid number of the original centrifugal fan discussed in this study is listed in Table 2. A general view of the geometry and grids is shown in Fig. 2.

Boundary conditions: Centrifugal fan geometry consists of three different zones: the inlet region, the impeller and the volute. The inlet region and the volute are considered as a stationary frame, whereas the impeller zone is studied as a rotating frame. Surfaces between the entry zone and the impeller (surfaces corresponding to the blades entry) ,and surfaces between the impeller and the volute (surfaces corresponding to the blades exit) are defined interfaces. In the two grid interfaces between a stationary and moving mesh, inlet-impeller and impeller-volute, the overlapping faces in the interface zones are determined as each new time step. Fluxes across each grid interface are calculated proportionally to the areas of the superposed faces. All of the wall boundary conditions in the volute and impeller are regarded as no-slip conditions.

Convergence of the solution is determined by monitoring the normalized residuals for each dependent variable. The solution is considered converged when these normalized residuals reduce to 1×10^{-3} and the mass residual decreases to 1×10^{-6}.

Mathematical models: For steady incompressible turbulent flows, the continuity and Reynolds-averaged Navier-Stokes equations are as follows:

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij} - \rho\overline{u_i' u_j'}\right]+ \bar{s}_i^u \tag{2}$$

where u_i and u'_i are mean and fluctuating velocities, respectively and $\bar{s}^u_i$ is source term. Governing equations with standard k - ε turbulence model are transformed to non-orthogonal curvilinear coordinates and are discretized with finite volume approximations. As a numerical scheme for the convection terms, a linear upwind differencing scheme is used and for the diffusion terms, a central differencing scheme is used. The Strongly Implicit Procedure (SIP) was used to solve linear algebraic equations. Also the SIMPLEC algorithm is used to match pressure and velocities.

RESPONSE SURFACE METHODOLOGIES

Response surface methodology is a set of mathematical and statistical techniques that are useful for modeling and predicting the response of interest affected by a number of input variables with the aim of optimizing this response 0028 (Bappa *et al*., 2012). RSM also specifies the relationships among one or more measured responses and the essential controllable input factors. The field of RSM consists of the following:

- Designing of a series of experiments for adequate and reliable measurement of the response of interest
- Developing a mathematical model of the second-order response surface with the best fittings
- Finding the optimal set of experimental parameters that produces a maximum or minimum value of response

- Representing the direct and interactive effects of process parameters through two-and three-dimensional plots

If all variables are assumed to be measurable, the response surface can be expressed as follows:

$$y(x) = f(x_1, \cdots, x_K) + \varepsilon$$

where, y is the response of the system and x_i is the variables of action called factors.

In the practical application of RSM, it is necessary to develop an approximating model for the true response surface. The approximating model is based on observed data from the process or system and is an empirical model. Multiple regression analysis is a collection of statistical techniques useful for building the types of empirical models required in RSM. Usually, a second-order polynomial equation is used in RSM:

$$y^P = c_0 + \sum_{i=1}^{K} c_i x_i + \sum_{i=1}^{K} c_{ii} x_i^2 + \sum_{j=1}^{K-1} \sum_{i=j+1}^{K} c_{ij} x_j x_i, P = 1 \cdots, n \tag{3}$$

where

y^p : Response (dependent variable)

c_0 : Constant coefficient

c_i,c_{ii}, c_{ji} : Coefficients for the linear, quadratic and interaction effec

x_i, x_j : Factors (independent variables)

The coefficients of determination R^2 and in Eq. (4) indicated the quality of the fit of polynomial:

$$R^2 = 1 - \frac{SS_{residual}}{SS_{model} + SS_{residual}} \tag{4}$$

In these equations, SS is the sum of squares.

EXPERIMENTAL DESIGN AND DATA ANALYSIS

The Design-Expert 8.0 software was used for the statistical design of simulations and data analysis. In order to ascribe the effect of factors on response surface in the region of investigation, a Box-Behnken design with three factors at three levels was performed. Investigated factors which were considered to have the greatest influence on efficiency were inlet blade angle, outlet blade angle, blade number.

Sample point design: First, inlet blade angle α, outlet blade angle β and blade number n were coded to (-1, 1), which is shown as Table 3.

Table 3: Design parameter and code

			Coded variable levels		
Parameter	Range	Code	-1	0	1
Inlet blade angle (α)	37~41	x_1	37	39	41
Outlet blade angle (β)	61~65	x_2	61	63	65
Blade number (n)	10~14	x_3	10	12	14

Table 4: The Box-behnken design and corresponding results (the responses)

No	Inlet blade angle x_1	Outlet blade angle x_2	Blade number x_3	Response: efficiency (%)
1	-1	-1	0	92.395
2	1	-1	0	91.755
3	-1	1	0	92.4214
4	1	1	0	92.4398
5	-1	0	-1	93.2409
6	1	0	-1	92.1942
7	-1	0	1	93.4758
8	1	0	1	92.7515
9	0	-1	-1	91.1253
10	0	1	-1	92.6324
11	0	-1	1	93.4941
12	0	1	1	92.7085
13	0	0	0	92.6665

In order to reduce the number of data needed for constructing response surface and to improve the representation of the design space, the Design of Experiment (DOE) is important for selecting design points. Among the different types of DOE techniques, the method of Box-Behnken is used to choose the sample point. The numerical simulation results are shown in Table 4.

Regression model and statistical analysis: The response obtained in Table 4 was correlated with the three independent variables used in the polynomial equation, Eq. (3). Least squares regression was used to fit the obtained data to Eq. (3). The final model, in terms of coded factors is expressed by the quadratic empirical Eq. (5):

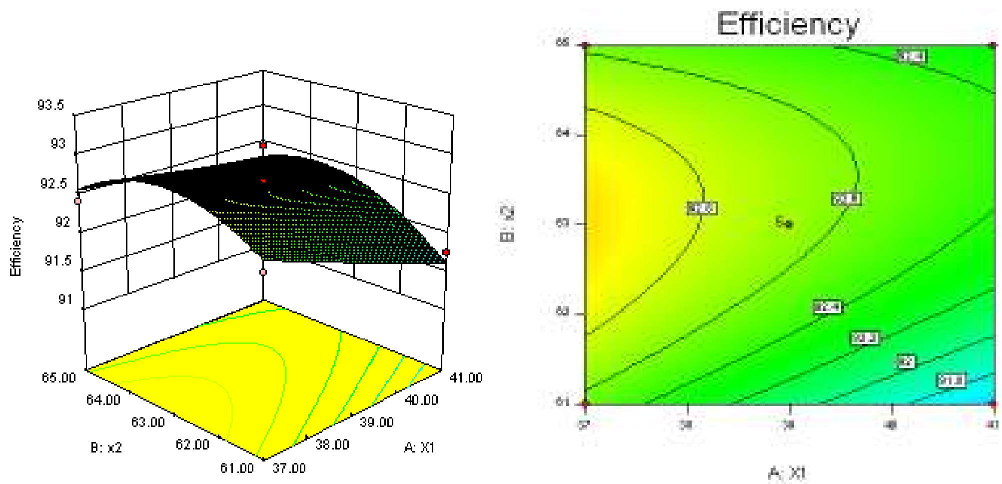
$$\eta = 92.67 - 0.3x_1 + 0.18x_2 + 0.4x_3 + 0.16x_1x_2 + 0.081x_1x_3 - 0.57x_2x_3 + 0.005912x_1^2 - 0.42x_2^2 + 0.24x_3^2 \tag{5}$$

ANOVA evaluations of this model, shown in Table 5, imply that this model can describe the simulations. To measure how well the suggested model fit the simulation data, the parameters F-value, R^2, p-value and lack of fit were used (Ceylan *et al*., 2008).

As we can see in Table 5, F-value was 7.18 and implied that the quadratic model was significant. Moreover, each term in the model was also tested for significance. R^2 = 90.23%, the adjusted determination coefficient is also high to prove the high significance of the model. From Table 5 it is clear that the linear terms for blade Number (x_3), the inlet blade angle (x_1) have

Table 5: ANOVA for the regression model and respective model terms

Source	SS	Degree of freedom	Mean square	F-value	p-value prob>F
Model	4.678314094	9	0.519812677	7.18	0.0082
x_1	0.715566845	1	0.715566845	9.89	0.0163
x_2	0.256578661	1	0.256578661	3.55	0.1017
x_3	1.309852051	1	1.309852051	18.10	0.0038
x_1x_2	0.10837264	1	0.10837264	1.50	0.2606
x_1x_3	0.02598544	1	0.02598544	0.36	0.5679
x_2x_3	1.314118323	1	1.314118323	18.16	0.0037
x_1^2	0.00014719	1	0.00014719	2.034E-003	0.9653
x_2^2	0.741366948	1	0.741366948	10.24	0.0151
x_3^2	0.249011201	1	0.249011201	3.44	0.1060
Residual	0.506580468	7	0.072368638		
Lack of Fit	0.506580468	3	0.168860156		
Pure Error	0	4	0		
Cor Total	5.184894561	16			



(a) (b)

Fig. 3: Response surface of efficiency as a function of inlet blade angle x_1 and outlet blade angle x_2

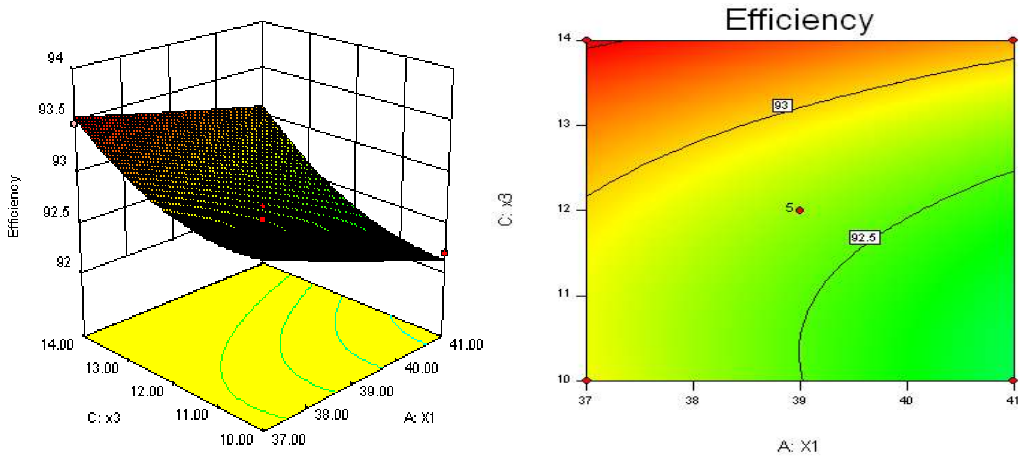
large effects on Efficiency due to high *F*-values. In addition, the linear term for outlet blade angle (x_2) is also significant but has a smaller effect on the efficiency due to its smaller *F*-value than the other linear terms. However, the quadratic term for the outlet blade angle (x_2^2) has a large *F*-value and a p-value =0.106. Thus, the effect of the outlet blade angle on the efficiency is most strongly modeled with the quadratic term. The quadratic term for blade Number (x_3^2) is also significant but has smaller *F*-values than their corresponding linear terms. The quadratic term for the inlet blade angle (x_1^2) is insignificant. The only significant coupling term is between outlet blade angle and blade number (x_2x_3), indicating an interaction between those two variables.

Influence of inlet blade angle and outlet blade angle: In order to investigate the interactive effect of inlet blade angle and outlet blade angle on the efficiency when number of blade is 12, the response surface methodology was used and 3-D surface response and contour plots were drawn (Fig. 3). The efficiency decreases when the inlet blade angle increases. The optimization inlet blade angle is about 37°. This plot implies that the operating conditions of 37° and about 63° lead to the maximum efficiency (over 92.8%). The efficiency also increases as the outlet blade angle increases from 61° to about 63° and thereafter decreases as the outlet blade angle increases from 63° to 65°.

Influence of inlet blade angle and blade number: Blade number has a positive effect on the efficiency. The quadratic term of this factor has a significant positive effect on the efficiency, but its value is smaller than the positive, linear term.

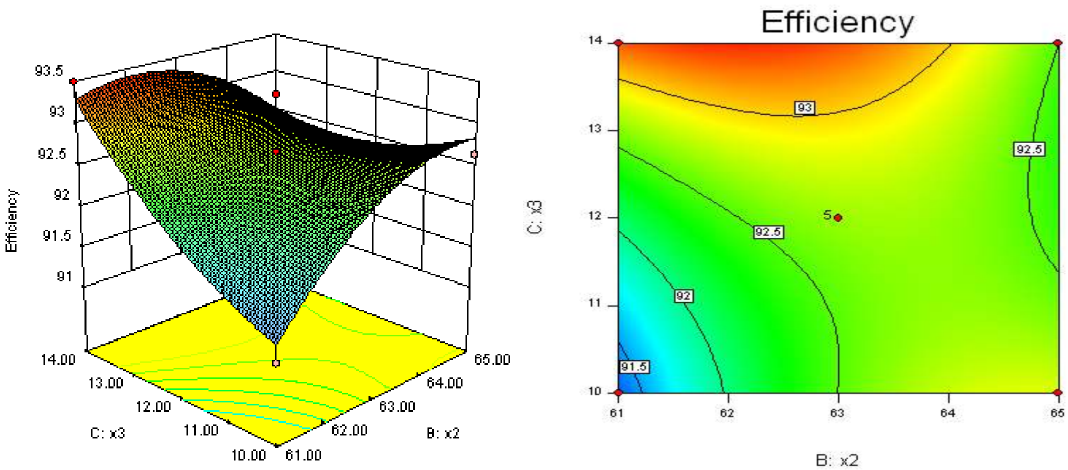
Figure 4 shows the influence of inlet blade angle and blade number on efficiency for the case when the inlet outlet blade angle is 63°. By increasing blade number, efficiency increases. The maximum efficiency which is about 93.5% is achieved when the blade number is 14 and the inlet blade angle is 37°. The inlet blade angle has a minor, linear, positive effect on the efficiency response. The quadratic influence of this factor has a small positive effect on the efficiency.

Influence of outlet blade angle and blade number: Figure 5 shows the influence of outlet blade angle and



(a) (b)

Fig. 4: Response surface of efficiency as a function of inlet blade angle x_1 and blade number x_3



(a) (b)

Fig. 5: Response surface of efficiency as a function of outlet angel x_2 and number of blade x_3

blade numbers on efficiency for the case when the inlet blade angle is 39^{o}. By increasing outlet blade angle, efficiency first increases and then decreases. The maximum efficiency which is about 93.5% is achieved when the blade number is 14 and the outlet blade angle is about $62^{o.}$

OBJECTIVE FUNCTION AND DESIGN VARIABLES

The following objective function is minimized in the optimization process:

$$F = 1 - \eta$$

The efficiency is defined as follows:

$$\eta = \frac{P_{t,ex} - P_{t,in}}{\rho(u_2 c_{2u} - u_1 c_{1u})}$$

where, P_t is total pressure and the subscripts, *in* and *ex* indicate the inlet and exit of the fan, respectively.

Figure 1 shows the design variables for impeller: inlet blade angle (α), outlet blade angle (β). Another design variable is the blade number.

The optimum factors derived via RSM were: inlet blade angle 37°, outlet blade angle 61.7^{o} and blade number 14. The maximum efficiency is 93.7%, under optimum conditions.

Based on the simulation method discussed above, the numerical calculation on the optimized fan was done. The RSM optimization result and CFD result are shown in Table 6.

From Table 6, we can get the message that RSM optimization method has higher prediction accuracy, the error of efficiency between RSM method and CFD calculation is 0.1 %. After optimization by RSM, the

Table 6: RSM optimization result and CFD result

	The optimized fan		
	RSM optimization result	CFD numerical calculation result	Original fan
Efficiency η	93.7%	93.6%	92.07%

efficiency of the fan improves and overall performance also improves significantly.

CONCLUSION

- The Response Surface Methodology (RSM) has been performed in order to study the effect of the impeller of a backward-curved centrifugal fan on the efficiency of the fan. Numerical simulation has been used to determine the efficiency of the fan for different configurations of the impeller parameters. These impeller parameters consist of inlet blade angle, outlet blade angle and blade number. Numerical results show that three parameters of the impeller have certain influence on the efficiency.
- Optimization results show that the RSM optimization method enables to improve aerodynamic performance of the original fan and that the response surface optimization result is very close to the corresponding CFD validation result.
- The combination of CFD with RSM can be applied to the optimization analysis of the centrifugal fan aerodynamic performance, which can not only provide a reference basis for the trial design, but also help to reduce the number of trials and the test cost. This research will provide some useful references for the fan energy-saving.

ACKNOWLEDGMENT

This study was financially supported by the program of Hebei dunshi engineering Technology Company (201017123).

REFERENCES

Bappa, A., S.K. Arunanshu, M. Souren and M. Dipten, 2012. Modeling and analysis of simultaneous laser transmission welding of polycarbonates using an FEM and RSM combined approach. Optics Laser Technol., 44: 995-1006.

Bayomi, N.N., A. Abdel Hafiz and A.M. Osman, 2006. Effect of inlet straighteners on centrifugal fan performance. Energ. Convers. Manage., 47: 3307-3318.

Behzadmehr, A., J.B. Piaud, R. Oddo and Y. Mercadier, 2006. Aero-acoustical effects of some parameters of a backward-curved centrifugal fan using DoE. HVAC R Res., 12(2): 353-365.

Ceylan, H., S. Kubilay, N. Aktas and N. Sahiner, 2008. An approach for prediction of optimum reaction conditions for laccase-catalyzed bio-transformation of 1-naphthol by Response Surface Methodology (RSM). Bioresour. Technol., 99: 2025-2031.

Kwang-Yong, K. and S. Seoung-Jin, 2006. Application of numerical optimization technique to design of forward-curved blades centrifugal fan. JSME Int. J., 49(1): 152-158.

Li, C., S.L. Wang and Y. Jia, 2011. The performance of a centrifugal fan with enlarged impeller. Energ. Convers. Manage., 52: 2902-2910.

Morinushi, K., 1987. The influence of geometric parameters on F.C. centrifugal fan noise. Trans. ASME J. Vib. Acoust. Stress, Reliabil, 109: 227-234.

Sheam-Chyun, L. and H. Chia-Lieh, 2002. An integrated experimental and numerical study of forward-curved centrifugal fan. Exp. Therm. Fluid Sci., 26: 421-434.

Son, P.N., J.W. Kim, S.M. Byun and E.Y. Ahn, 2012. Effects of inlet radius and bell mouth radius on flow rate and sound quality of centrifugal blower. J. Mech. Sci. Technol., 26(5): 1531-1538.

Zhao, Y., L. Song, H. Wenqi, W. Weixiong, H. Dongtao and Z. Zhichi, 2005. Numerical simulation of flow field for a whole centrifugal fan and analysis of the effects of blade inlet angle and impeller gap. HVAC R Res., 11(2): 263-283.