VOL. 9, NO. 9, SEPTEMBER 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

DESIGN OPTIMIZATION OF CENTRIFUGAL FAN OF TRAVELLING CLEANER

C. N. Jayapragasan, Sumedh J. Suryawanshi and K. Janardhan Reddy
SMBS, VIT University, Chennai Campus, Chennai, India
E-Mail: jayapragasan.cn2013@vit.ac.in

ABSTRACT

Centrifugal fans play an important role in the proper functioning of any travelling cleaner. This study presents a design methodology to examine the performance of the fan using computational fluid dynamics approach. The effect of fan geometry, fan speed and fillet radius at the inlet on performance of the fan have been assessed. Number of blades and the volute dimensions has been kept constant. Total discharge and fan total efficiency are the output parameters calculated. In order to reduce the number of trails, Taguchi method is used. The fan is modeled using Solid Works 2012 and after simplification the modeled fan is meshed in ICEM CFD. The solution is obtained using FLUENT V6. The post processing is carried out using CFD POST and the results are presented and discussed in detail. Finally the using Minitab 16.0 the responses of parameters have been plotted and the optimum values of the parameters are obtained. These obtained values need to be implemented into the design for better performance of the fan.

Keywords: centrifugal fan, computational fluid dynamics (CFD) analysis, Taguchi method.

1. INTRODUCTION

In many industries, travelling cleaners are used to keep clean the working environment and machines. The cleaner with the help of fan sucks unclean air around the machine and then the solid particles in the air are filtered and removed by manually or centralized collection system. The filtered air is then discharged at high velocity and pressure on the machines and floor, thus keeping the machine and floor also clean. The performance of this industrial cleaner is mainly dependent on the fan used. Hence there arises a need to study the performance parameters of the fan.

The most common fan used in travelling cleaner is centrifugal fan. A fan mainly consists of four major parts as seen in the Figure-1.

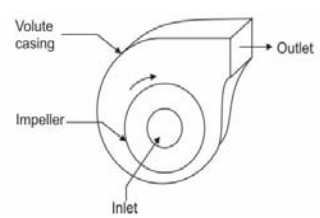

Figure-1. Parts of a centrifugal fan [1].

Centrifugal fans being a mechanical device transfer air and other gases. The blades in the impeller of the fan are fixed between the inner and outer diameters. Through the inlet the air enters the impeller axially which provides slight acceleration to the air before entering the impeller. These impellers rotate at very high speeds to move air using centrifugal action radially outwards which is then passed into a spiral shaped casing often called as volute casing. Then the air moves in tangential direction away from the blade tips of the fan. The casing increases the static pressure of the air and finally discharges the air from the outlet.

The curvature of centrifugal fan blades is almost similar to the aero-foil cross section and thus provides better efficiency at a very economical value. These centrifugal fans are often used for high pressure, high power and medium flow applications [1] The Figure-2 shows the arrangement of the blades in a centrifugal fan.

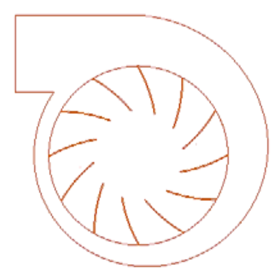

Figure-2. Arrangement of blades in centrifugal fan.

For any centrifugal fan, the geometry and the fan speed play an important role in the performance of the fan. Hence a comprehensive study is carried out in order to understand the influence of different design factors on the performance of the fan. This study focuses on design of a fan to cater the need of compact in size and deliver the required effort with better performance. Keeping the some parameters constant, like the tip clearance, number of blades, parameters like fan speed, impeller outer diameter, fillet at the entry to the impeller are varied to get an optimum design reducing the size of the fan with same or increase in the performance of the fan.

2. NOMENCLATURE

Nomenclature used in this study.

CFD	Computational Fluid Dynamics
D	Outer diameter of impeller, mm
MRF	Moving Reference Frame
N	Fan speed, rpm
P	Power of fan, kW
P_{TF}	Total pressure developed by fan, Pa

VOL. 9, NO. 9, SEPTEMBER 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

Q	Discharge, cfm
S/N ratio	Signal to noise ratio
cfm	Cubic feet per meter
mm	millimeters
mm in WC	Millimeters of water column
rpm	Rotations per minute

3. LITERATURE SURVEY AND RELATED WORK

The performance of the fan is governed by different fan laws. Hence lot of emphasis is given to understand the basic theory of fans, their types and their working. The selection of critical parameters is very essential while determining the performance of the fans [1]. The basic equations mainly continuity equation, momentum equations and energy equations need to be considered while following the computational fluid dynamics approach. While considering any practical problem the best turbulence model and order of accuracy needs to be selected for good results [2].

When carrying out the design optimization of centrifugal fans, a complete design methodology is to be followed. It is also necessary to develop a numerical procedure whose results can be compared with the results from CFD simulation [3].

While considering the different parameters and their interface with each other, the number of trials required is more. This also increases the computational time and computational cost. To reduce the number of trails to be conducted Design of Experiments (DOE) approach can be implemented. Taguchi orthogonal array can be easily used to reduce the number of trails. It also helps use to select the optimum values to get maximum results. Use of Minitab software facilitates faster and easy calculation [4].

The fan to be analyzed is modeled using CAD software and then it is meshed. The fan needs to be modeled correctly and it needs to be simplified for ease of meshing by deleting unwanted surfaces. The selection of meshing technique plays a vital role in analysis. The different mesh parameters are varied to get the best meshing possible [5]. The flow is then simulated by using the popular RANS (Reynolds Averaged Navier Stokes) equation with a proper turbulence model [6].

For any fan, the weight, size and noise vibrations directly affect the performance of the fan. So without compromising on the quality, fan can be redesigned to reduce these factors using finite element approach [7]. Any simulations results are not valid unless compared with the experimental results. The fan is tested experimentally with same boundary conditions. Parametric studies are carried out to compute the power coefficient along with flow coefficient and efficiency. The parameters considered are number of blades, outlet angle and diameter ratio. Finally a systematic and reliable strategy is required to correctly interpret the CFD solutions and compare them with the experimental results [8].

While performing the experimental and numerical study the essential parameters to be determined are detailed flow visualization, torque calculation, efficiency estimation and noise analysis. The results are fairly accurate using to solve 3D Navier-strokes equation and adopting the second-order upwind scheme for k - ε calculations [9].

Hence in this study a fan is selected and its performance is calculated using CFD approach. Different parameters will be analyzed and various modifications will be implemented for the analysis to improve the performance of the fan.

4. METHODOLOGY AND ITS IMPLEMENTATION

For this study of centrifugal fan, the methodology followed comprises of following steps:

a) Literature survey
b) Identification and selection of parameters to be studied
c) Implementation of Taguchi techniques for deciding the number of trails
d) Modeling, meshing and analysis of the fan
e) Mathematical validation
f) Result comparison and selection of best combination

4.1. Identification and selection of parameters

The main parameters identified for this particular fan are impeller diameter, fan speed and fillet radius at the inlet. The main purpose of this study is to study the combined effect of these parameters on the performance of the fan along with their importance.

4.2. Implementation of Taguchi method

In present study we have to focus on three major parameters having two levels each. Hence we can use the standard Taguchi L4 orthogonal array to decrease the number of trails. The parameters with their levels are presented in Table-1.

Table-1. Parameters with their levels.

Parameters		Levels	
		1	2
A	Impeller outer diameter, mm	382	372
B	Fillet radius, mm	2	10
C	Fan speed, rpm	2800	3450

Using the values of parameters and levels the L4 orthogonal array is given in Table-2. As seen in the Table only 4 trials are required to understand the effects of the parameters.

Table-2. L4 Orthogonal array.

Experiment	A	B	C
1	382 mm	2 mm	2800 rpm
2	382 mm	10 mm	3450 rpm
3	372 mm	2 mm	3450 rpm
4	372 mm	10 mm	2800 rpm

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

4.3. Modeling of centrifugal fan

The three dimensional centrifugal fan model is created using SolidWorks 2012. The design of centrifugal fan mainly include: Impeller outer diameter, Blade Thickness and Impeller inlet diameter. Geometry parameters were changed based on the Taguchi OA. The specifications of the present centrifugal fan modeled are summarized in Table-3.

Table-3. Design specifications.

S. No.	Parameter	Original fan
1	Impeller outer diameter, mm	382
2	Impeller inner diameter, mm	250
3	Impeller width, mm	68
4	Number of blades	12
5	Blade thickness, mm	2
6	Inlet area, m^2	0.020
7	Outlet size, mm x mm	164.80 x 84

The Figure-3 and Figure-4 show the dimensions of casing of the centrifugal fan and the blade construction respectively.

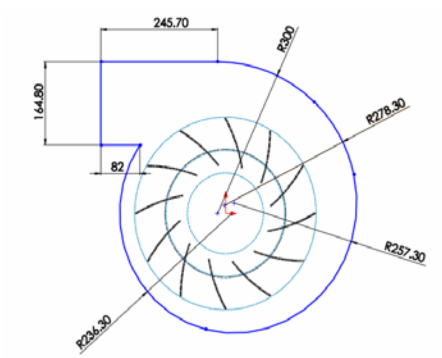

Figure-3. Dimensions of the casing.

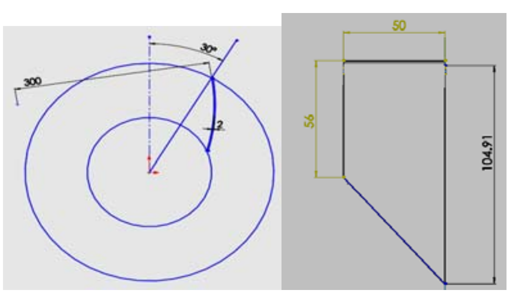

Figure-4. Blade geometry and its construction.

Along with these dimensions, a fillet radius of 2 mm and 10 mm is given for two different fans and there effects are observed on the performance of the fan. Figure-5 shows the fillet radius considered.

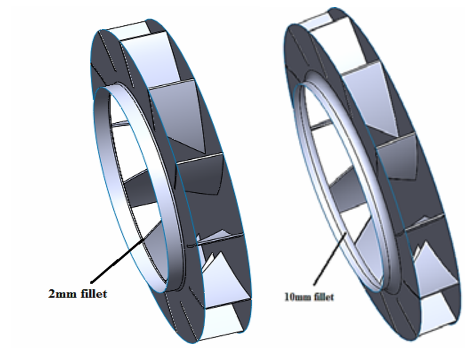

Figure-5. Fillet radius of 2mm and 10mm at inlet to the impeller.

The final 3D modeled fan is given in Figure-6.

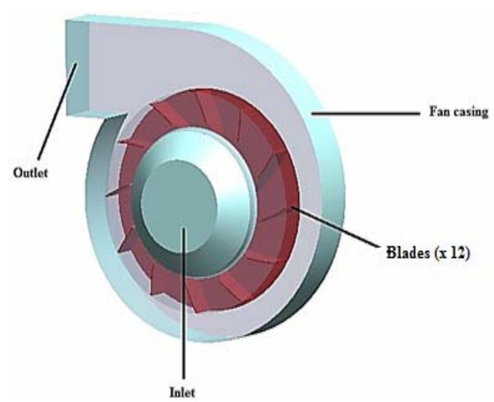

Figure-6. 3D model of fan.

4.4. Meshing of 3D model

After modeling the fan, the parts are simplified for sake of simplicity in meshing. Two fluid domains are modeled one around the impeller and other one in the casing. The 3D CAD model is imported into ICEM CFD software for meshing. The models are meshed using tetrahedral elements as they are easy to use for complex geometries as compared to other type of elements. Parts of prime importance in this study are fine meshed as compared to other parts. After successful grid generation, mesh independency test were carried out before moving onto the simulation. Test grid independency test results are numerically shown in Table-4.

From the Table-4 it is clearly evident that mesh b and mesh c has almost same values. Hence mesh b is selected to reduce computational time. Figure-7 shows the meshed 3D model.

Table-4. Mesh independency test.

Mesh	Number of mesh elements	Mass flow rate kg/s
a	743982	0.542
b	1791892	0.677
c	2063458	0.691

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

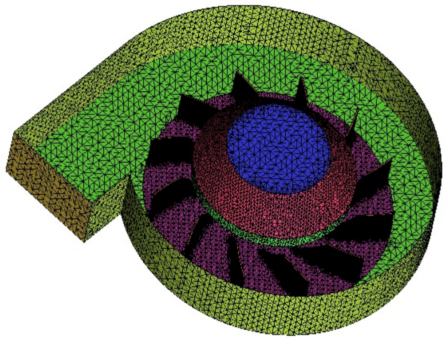

Figure-7. 3D meshed model of fan.

4.5. Computational approach

Computational fluid dynamic (CFD) approach is one of the effective methods of solving the fundamental non-linear PDE equations that rule the fluid flow, heat transfer and turbulence of flow. For the existing work, CFD simulations are performed using the commercial CFD software package FLUENT V6. This software solves the Navier Stokes equation governing the laws of physics of flow inside centrifugal system.

In problems comprising rotating motion FLUENT V6 adapts a method known as moving reference frame. The moving reference zone is a fluid domain created around the moving parts of a system. For centrifugal fan system the MRF is rotating domain. The faces of the MRF are specified as internal faces. The steady state approximation used in MRF model, permits individual cell zones to rotate or translate with variable speeds. MRF model is used in fans as the rotating member. This is attained by dividing the domain into distinct zones where the fluid flow is solved in stationary or rotating coordinate systems.

For this particular study the assumptions made are:

a) Steady state air flow
b) Segregated solver and implicit formulation
c) Standard wall functions
d) Turbulence Kinetic Energy-Second Order Upwind Scheme
e) Momentum-Second Order Upwind Scheme

4.6 Boundary conditions and case setup

The boundary conditions along with case setup are summarized in Table-5.

Table-5. Boundary conditions and case setup.

Boundary condition	Value
Material	Air
Density	1.225 kg/m^3
Viscosity	1.789e^{-05} kg/m-s
Turbulence model	K-ω SST model
MRF	Impeller
Inlet condition	Pressure inlet
Outlet condition	Pressure outlet
Pressure at inlet	200 mm in WC
Pressure at outlet (Pa)	0
Hydraulic diameter at inlet (m)	0.160
Hydraulic diameter at outlet (m)	0.110
Solution of Navier Stokes Equation	SIMPLE algorithm
Impeller speed (rad/s)	293.21 361.28

4.7. Post processing and results

Once the solution gets converged the results are post processed using CFD POST. Various values like total pressure, static pressure, velocity are obtained.

4.8. Mathematical validation

The theoretical values of efficiency for the present fan are calculated using the fan formulae [10]. The fans laws are used for calculating the fan power and efficiency for modified fans. The power consumption for electric motor was selected as 2.2KW and 3.7 KW respectively. The basic fan laws are summarised below:

$$Q \propto N \propto D^3 \quad (1)$$

$$P_{TF} \propto N^2 \propto D^2 \quad (2)$$

$$P \propto N^3 \propto D^5 \quad (3)$$

The values obtained using above fan laws are further compared with values from CFD simulations.

5. RESULTS AND DISCUSSIONS

5.1. Post processing results

The different plots for various cases are depicted below.

5.1.1. Trail 1

the original fan, Figure-8 shows the velocity contours, pressure contour plotted on mid-plane respectively.

VOL. 9, NO. 9, SEPTEMBER 2014 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

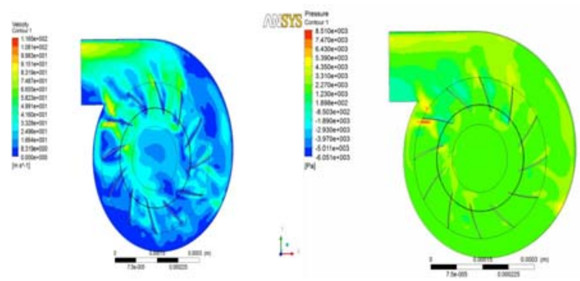

Figure-8. Velocity contour and Total pressure at the mid plane.

From above figure it can be seen that there is gradual rise in velocity and pressure as the fluid flows from inlet to outlet. Figure-10 shows the velocity vectors near the blade nearest to the outlet. It can be observed that some of the fluid directly bypasses from the impeller side towards the outlet.

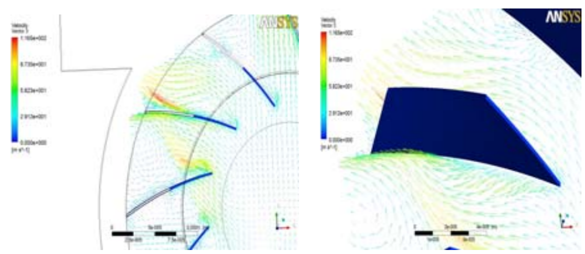

Figure-9. Velocity vectors on blade nearest to the outlet.

5.1.2. Trail 2

Figure-10 shows the velocity contours and pressure contours. As in this trail the fan speed has increased it can be observed there is an increase velocity of fluid from inlet to outlet.

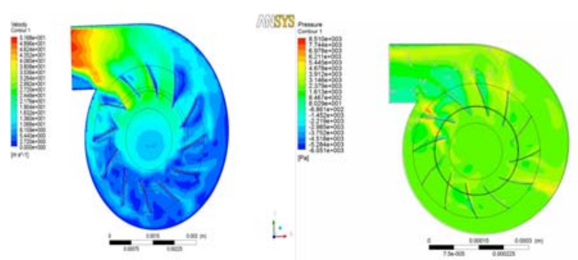

Figure-10. Velocity contour and pressure contour at the mid plane.

5.1.3. Trail 3

The velocity and pressure contours for trail 3 are shown in Figure-11.

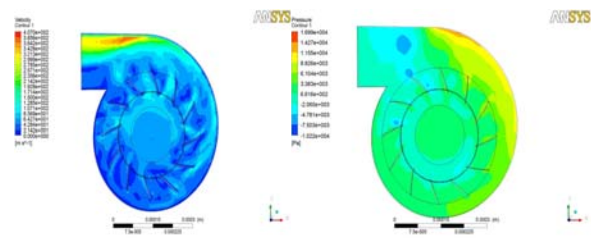

Figure-11. Velocity contour and pressure contour at the mid plane.

5.1.4. Trail 4

The velocity contour and pressure contour at the mid-plane are given in Figure-12.

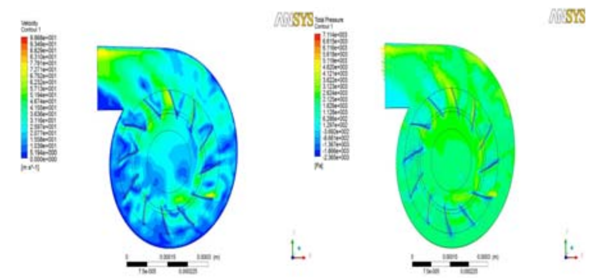

Figure-12. Velocity contour and pressure contour at the mid plane.

5.2. Effect of fillet radius at the entry to the impeller

In this study an attempt had been made to understand the effect of any fillet radius at the entry to the impeller. The fillet radius given is 2mm and 10 mm respectively. The velocity vectors and velocity contours for both the fillet radius are given in Figure-13 and Figure-14.

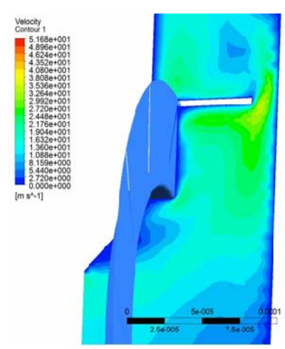

Figure-13. Velocity vector and velocity contour at the entry to the inlet with the fillet radius 2mm.

It can be observed from Figure-13 and Figure-14 that the velocity is increased with modification at the entry to the impeller.

ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

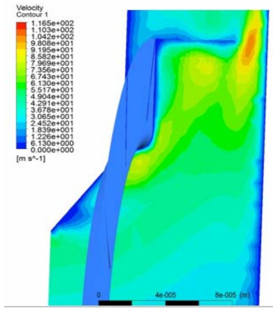

Figure-14. Velocity vector and velocity contour at entry to the inlet with fillet radius 10mm.

5.3. RESULT TABLE OF TAGUCHI METHOD

Minitab 16.0 software is used to understand the effect of parameters and to get the optimized values for maximum discharge and efficiency. The response analysis is carried out by results from CFD simulations corresponding to the selected parameter and level in Table-6. Figure-15 and Figure-16 give the results of mean effects plot for means and for SN ratios.

The impeller outer diameter affects the performance of the fan the most while the fillet radius at inlet affects the least as derived from the means as in Table-7. Since our main aim is to maximise the efficiency and discharge of the fan, the larger the better concept is used to select the optimum values of the parameter. The optimum values of different parameters are summarised in Table-8.

Table-6. Trails and the results.

Trial	**Impeller outer diameter** (mm)	**Fillet radius at the entry of impeller** (mm)	**Fan speed** (rpm)	**Discharge from CFD results** (cfm)	**Fan total efficiency from CFD results** (%)	**Fan total efficiency theoretical** (%)
1	382	2	2800	1177	57.68	59.86
2	382	10	3450	1496	63.47	69.23
3	372	2	3450	1287	59.31	61.54
4	372	10	2800	1046	49.86	55.34

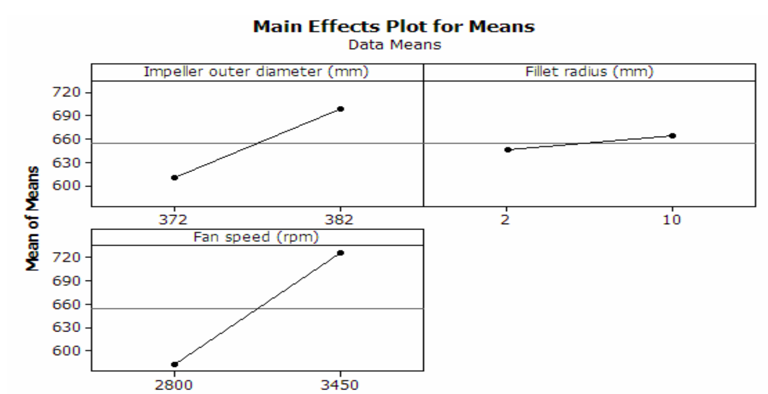

Figure-15. Responses for parameters by mean effect plots for means.

Table-7. Maximum affecting parameters.

Levels	**Impeller outer diameter** (mm)	**Fillet radius at the entry of impeller** (mm)	**Fan speed** (rpm)
1	614.34	647.14	683.69
2	702.85	662.37	733.89
Δ	88.51	15.23	50.2
Rank	1	3	2

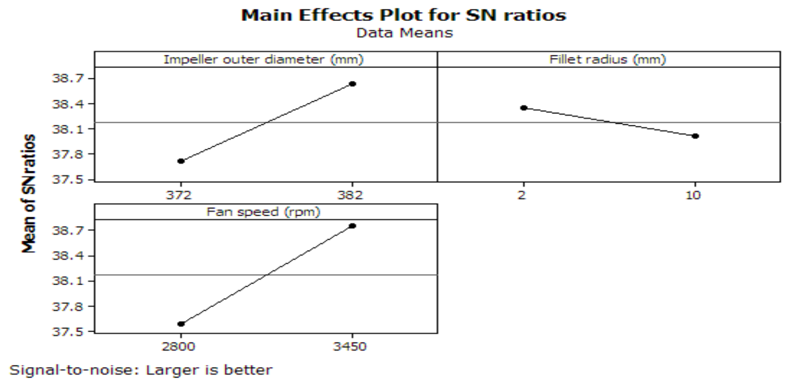

Figure-16. Responses for parameters by S/N ratios.

Table-8. Optimized input values.

S. No.	**Impeller outer diameter** (mm)	**Fillet radius at the entry of impeller** (mm)	**Fan speed** (rpm)
1	382	2	3450

6. CONCLUSIONS

This paper studies in detail the optimization of centrifugal fan using computational fluid dynamics approach. The major parameters like the impeller outer diameter and fan speed are considered. The effect of fillet radius at the entry of inlet is observed on the performance of the fan. The CFD approach helps to improve the results. However some variations are observed in numerical results using CFD and theoretical results.

This may be due various assumptions considered in numerical procedure and CAD modelling. Taguchi method thus has helped to reduce the number of trials and save computational time with giving the optimum result. The results of this study will surely help to improve the performance of the fan.

7. FUTURE SCOPE OF THIS RESEARCH

There is a vast scope for this research in order to improve the design of the centrifugal fan. In future more parameters affecting the performance of the fan can identified, selected and studied in detail. The shapes and orientation of blades used can be varied to understand the effects on fan performance. The mesh generated can be refined to get more accurate results. By considering more parameters and levels the number of trails can be increased to get better results. Small modifications as change in fillet radius can be verified to observe the effect of flow of fluid in the centrifugal fan by considering more number of values.

The performance of fan at different time conditions can be observed by considering transient state analysis. The optimized values obtained in this study need to be used to calculate the fan performance. Finally the results obtained using the numerical results need to be verified experimentally. If the results are satisfactory, the modifications can be implemented in the present fan giving better output and efficiency.

ACKNOWLEDGEMENT

Special thanks go to Dr. Sivakumar R. for his guidance in CFD analysis.

REFERENCES

[1] Yahya S M. 1998. Turbines Compressors and Fans. Tata Mc-Graw Hill Edition.

[2] Anderson J.D. 2012. Computational fluid dynamics. Tata Mc-Graw Hill Edition.

[3] Atre PC, Karuppa TR. 2012. Numerical Design and Parametric Optimization of Centrifugal Fans with Aerofoil Blade Impellers. Research Journal of Recent Sciences. 1(10): 7-11.

[4] Singh R R *et al*. 2012. Optimizing impeller geometry for performance enhancement of a centrifugal blower using the taguchi quality concept. International Journal of Engineering Science and Technology (IJEST). 4(10).

[5] Chand P, Rai S. 2012. Centrifugal fan impeller design with optimization of blade. International Journal of applied engineering research. 7(11).

[6] Dang Q. 2007. Performance improvement of centrifugal fan by using CFD, Hindawi Publishing Corporation. International journal of rotating machinery. Vol. 2007.

[7] Mohaideen M. 2012. Optimization of Backward Curved Aerofoil Radial Fan Impeller using Finite

Element Modeling. Procedia Engineering. 38: 1592-1598.

[8] Lin SC, Tsai ML. 2012. An integrated performance analysis for a backward-inclined centrifugal fan. Computers and Fluids. 56: 24-38.

[9] Singh OP. 2011. Parametric study of centrifugal fan performance: experiments and numerical simulation, IJAET.

[10] Gupta R S. 2010. Refrigeration and air conditioning, S. Chand.