PARAMETRIC STUDY OF CENTRIFUGAL FAN PERFORMANCE: EXPERIMENTS AND NUMERICAL SIMULATION

O. P. Singh[1*], Rakesh Khilwani[2], T. Sreenivasulu[1], M. Kannan[1]
[1]TVS Motor Company, R&D Department, Hosur, Tamilnadu-635109
[*]omprakash.singh@tvsmotor.co.in; om.prakashh.singh@gmail.com
[2]Department of mechanical engineering, Indian Institute of technology, Madras- 600036

ABSTRACT

In this paper, effect of geometric parameters of a centrifugal fan with backward- and forward-curved blades has been investigated. Centrifugal fans are used for enhancing the heat dissipation from the IC engine surfaces. In the process, the fan consumes power generated from the engine. As a first step, an experimental setup was developed and prototypes of fans were made to carry out measurements of flow and power consumed by the fan. The fan mounting setup was such that fan with uniform blades can be tested. Generally, fans have cut blades on the vehicle due to mounting accessories. We describe a patented design of shaft that would enable mounting of fan with uniform blades on the vehicle. Next, a computational fluid dynamics (CFD) model was developed for the above setup and the results are validated with the experimental measurement. Further, parametric studies were carried out to quantify the power coefficient, flow coefficient, efficiency and flow coefficients. The parameters considered in this study are number of blades, outlet angle and diameter ratio. The results suggest that fan with different blades would show same performance under high-pressure coefficient. However, the difference between the performances becomes distinct under low pressure coefficients suggesting that the fan performance testing should not be done on vehicle level where high pressure coefficients is observed due to various resistances in the system. The results show that increase in flow coefficient is accompanied by decrease in efficiency and increase in power coefficient. Effect on the vehicles mileage due to the use of forward and backward fan is also discussed. In summary, this study presents a systematic and reliable strategy to investigate the centrifugal fan performance in automotive applications.

KEYWORD

Centrifugal fan, Fan performance, Experimental validation, CFD, Automotive engine cooling

1. INTRODUCTION

Centrifugal fans are widely used in various fields of engineering [1-6]. In automotive industries, fans are used for cooling internal combustion engines [7, 8]. With increasing demands of high performance engines, cooling requirements have also increased proportionately to keep the engine metal temperatures within the desirable limits. Using an optimized fan for engine heat dissipation is critical to the overall engine performance [9, 10]. The fan derives its energy from the power generated by the engine. The fan has to overcome high system resistances when used in the vehicle level testing.

NOMENCLATURE & ABBREVIATION

A Inlet area, mm^2
P Pressure, pa
p_t Total pressure difference , pa
Q Volume flow rate, m^3/s
N_b Number of fan blades
Φ_0 Blade Outlet angle
Φ_I Blade Outlet angle
r_0 Outer radius of the fan, mm
r_i Outer radius of the fan, mm
d_r Diameter ratio
k Turbulence kinetic energy, m^2/s^2

u Mean velocity, m/s
T Fan torque
v Velocity, m/s
$\vec{u}_r$ Relative velocity, m/s
α Inlet angle, degrees
β Outlet angle, degrees
φ Volute angle, degrees
δ Kronecker delta
ε Dissipation rate, m^2/s^3
ρ Density, kg/m^3
$\bar{\bar{\tau}}$ Shear stress, N/m^2

μ Dynamic viscosity of air, kg/m-s
RPM Revolution per minute
CAD Computer aided design
CFD Computational fluid dynamics

Definitions

Flow coefficient $\dfrac{Q}{2\pi r^2 \dfrac{\pi r N}{30}}$

Pressure coefficient $\dfrac{P_t}{\rho\left(\dfrac{\pi r N^2}{30}\right)}$

Efficiency $\dfrac{P_t Q}{T\omega}$

Power coefficient $\dfrac{Flow\ coefficent\ X\ Power\ coefficent}{Efficiency}$

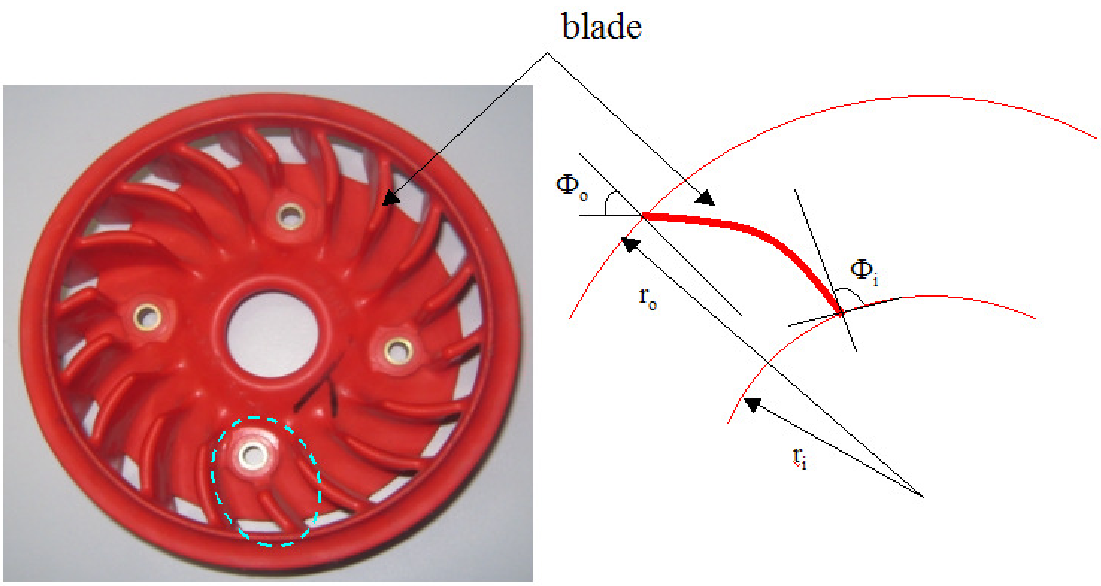

Figure 1 (a). Backward curved blade fan showing how blades are cut due to mounting accessories (b) line diagram depicting fan parameters.

In automotive industry, one way to test the fan performance for engine temperature reduction is to directly mount the fan on the engine cooling system [11, 12]. Engine is then run at desired speed for a specific time. Temperatures of various engine components are measured till the steady state is reached [13, 14]. However, this method has various disadvantages. First, since the system resistance is high, fan with various design configurations cannot be evaluated properly. Second, it is a time consuming process. Third, it requires the whole engine cooling system to be in place. Hence, under such conditions, effect of different fan parameters like number of blades, diameter ratio, outlet angles etc. cannot be evaluated properly. In this paper we show that true performance of a fan cannot emerge if tested in a high resistance system. Fourth, fan design as per drawing specifications and fan design that is used in testing is generally different. Reason being that screws and washers are used to mount the fan on engine. Few blades of fan are cut (see Fig. 1a) to accommodate these mounting accessories. However, blades are uniform as per design specification. Cut blades effects fan performance when compared to uniform blades. In the experimental setup described here, blades remain uniform due to the innovative mounting system (a patent has been filed). Results are compared that are obtained from the parametric study in experiments and CFD simulations with uniform blades.

One concern in automotive industries is the power consumption and efficiency of forward and backward curved blade fans. It is generally reported that forward curved fan have low efficiency and consumes more power generated by the engine [15, 16]. Hence, dilemma exists for the kind of fan to be used for the given specifications of the engine. In this paper we have quantified the efficiency and power consumption by these fans and tested on the engine for change in vehicles mileage. It was observed that even with 42% higher power consumption and 4.5% low efficiency of the forward curved fan, drop in mileage was insignificant.

Further, the number of fan blades needed for a given engine cooling system cannot be determined theoretically and it can be found only experimentally [17]. Hence, there is a need for a study that provides the quantitative effect of blades on power coefficients, flow coefficients, pressure coefficient and efficiency [18]. In this paper we present an experimental setup and a CFD model to study a centrifugal fan. Effect of various fan parameters is investigated at various operating conditions.

The paper is organized as follows: we describe the experimental setup and measurement method in section 2. CFD model and validation of results with the experiments are reported in section 3. Results and discussions are presented in section 4 followed by conclusions in section 5.

2. EXPERIMENTAL SETUP

In this section, we briefly describe the design of volute, casing, shaft and fan design. Main purpose of this experimental setup was three fold: a less resistance fan test setup, a shaft design that enables mounting of uniform blade length of the fan, and precise measurement of fan power consumption. These factors will enable us to perform one-to-one validation of the CFD model and to conduct further parametric investigation of fan performance.

2.1. Volute and casing design

The centrifugal fan is surrounding by a volute, which along with the sidewalls form a part of the casing. The volute along with cowling used in the vehicle has a very complex design with high resistance owing to the vehicle and engine design [13]. Hence, it is very difficult to test the fan in the actual engine cooling system. The volute profile of a fan mainly depends upon its outlet diameter and angle of absolute velocity with the peripheral direction [17]. The profile of volute is given by,

$$r = r_0 e^{\varphi \tan\alpha_0} \qquad (1)$$

Here r denotes the radius of the volute at an angle φ, r_0 is the outer radius that is equal to 66 mm in this case. α_0 is the angle that absolute velocity vector makes with the peripheral direction. Outer radius was kept same for all fan design. We have taken an average outlet angle α of 30°. For better performance measurement of fan and to reduce losses due to re-circulation, the clearance between the casing and fan was kept 5mm.

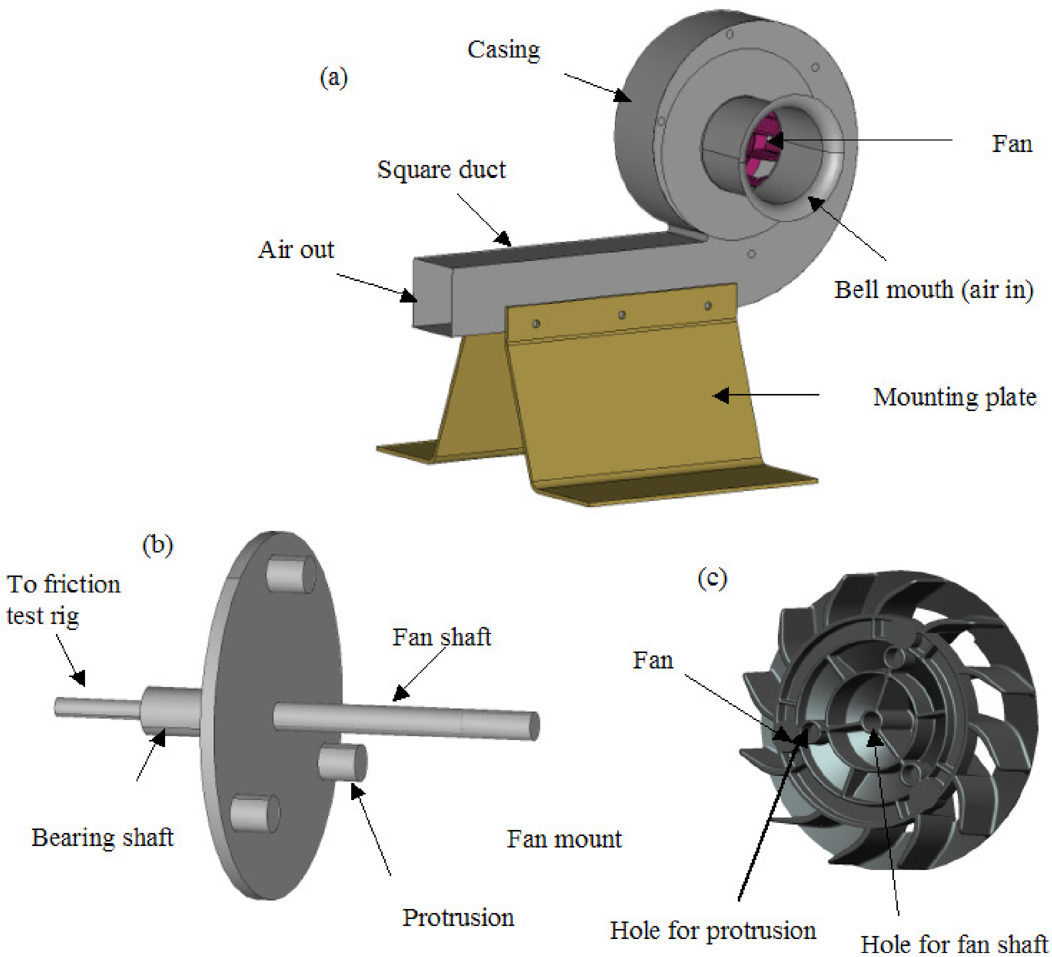

Figure 2(a). CAD model for the experimental setup (b) mounting shaft for fan and (c) fan.

Fig. 2(a) shows the casing design made of sheet metal of 2 mm thickness. On the back wall where bearing is required to be placed in the fan shaft, a thickness of 4 mm was provided for system stability reasons. Clearance between the casing and the fan from the sidewall = 7mm. This clearance was given in order to accommodate the shaft required to transmit the drive from the fiction test rig to the fan.

A bell shaped inlet duct of radius 2 mm is provided in order to reduce losses at the entry of the fan. Width of casing = 49 mm. A ball bearing with specification N6000 was used to mount the fan shaft on the casing. The ball bearing was press fit into the casing on the back wall. The shaft that transmits drive has one end fixed to the friction test rig coupling and the bearing supports the other end in order to prevent the bending of the shaft.

The heights of the two plates mounted at the bottom of the casing are such that the axis of fan and friction rig coincides. This is very critical for accurately measuring the power consumed by the fan. The plate was bolted to the friction rig test bed. This provides rigid supports for the casing even at high rpm when the vibrations in the system are very high.

2.2. Shaft and fan design

The design of shaft and fan is shown in Fig. 2(b,c). The end of the shaft, which connects to the test rig coupling, is made smaller in diameter compared to the shaft portion, which accommodates ball bearing due to stability reasons. Three protrusions are provided on the shaft, which accommodates the fan. This restricts the fan from the rotational motion. The longest part of the shaft goes inside the fan center and screwed at the end. This restricts the axial motion of the fan. The innovative part of this shaft is that it maintains the uniform length of the blades. This shaft serves the purpose of engine crankshaft. In engine assembly, the crankshaft holds the magneto, which in turn holds the fan. This design of the shaft has been patented. In prior art, fan is mounted on the magneto with screws, spring washers and other accessories. This results in cutting of the few blades (Fig. 1a), which affects fan performance. Following are the dimensions of the shaft: shaft length that goes into the friction rig = 50 mm, diameter at the bearing = 10 mm, length= 100 mm, shaft diameter at the fan = 6 mm and length= 150 mm, pitch circle diameter of the protrusions = 60 mm, diameter of protrusions = 7.9 mm, length = 8 mm. Hole diameter of the fan that accommodates protrusion: 8.1 mm. The casing and shaft was made in-house whereas fan was procured from supplier as per our design specifications.

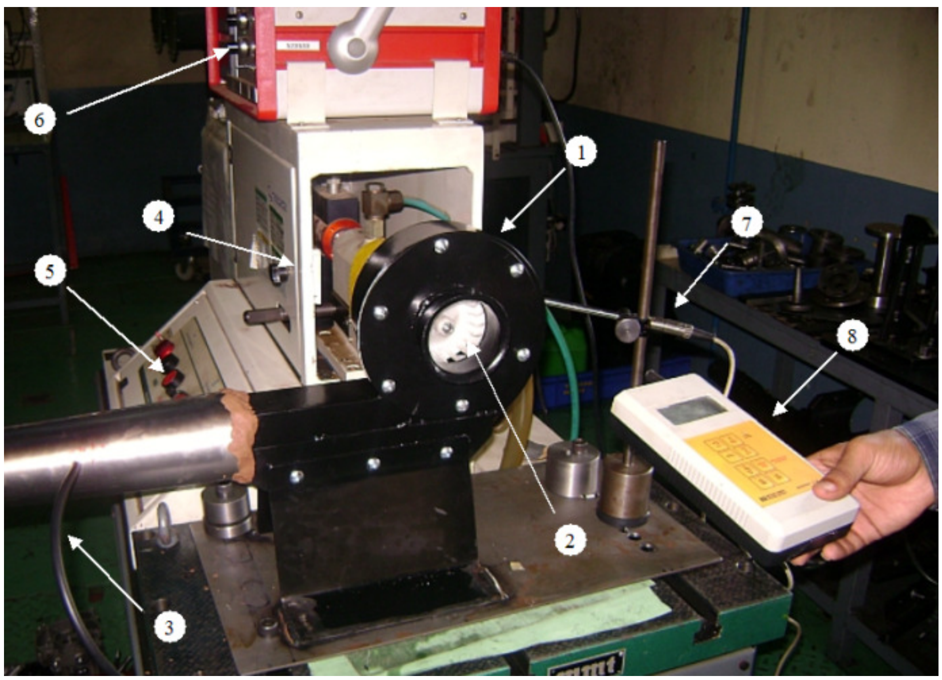

Figure 3. Developed experimental setup for fan testing. The numbers in the figure denotes: (1) volute casing, (2) fan (3) tube connected from volute outlet to the U-tube manometer, (4) low friction torque measurement machine (5) RPM control switch, (6) torque value displayer unit (7) optical sensor to measure rpm (8) digital meter for rpm display.

2.3. Measurement method

Fig. 3 show the experimental setup mounted on the low friction test rig. Different equipment used to measure fan rpm, fan torque and static pressure is shown. The rpm of the test rig can be varied from 0 to 8,500. During the first test at high rpm of about 6000, the few fan blades were observed to have developed cracks at the root. Due to components safety concerns fan data was measured up to 5000 rpm only. An LCD display on top of the test rig displays the torque values. The least count of rpm measuring instrument was 1 rpm and that of friction test rig was 0.001 Nm for torque measurement. The maximum possible torques that could be measured is 0.1 Nm and maximum misalignment of the shaft from the axis can be 0.1 mm. The design of the casing, shaft fan are such that these two limits were not exceeded.

Velocity at the outlet of the duct was measured with a vane type wind velocity meter. The meter gives average velocity at the cross-section. Advanced measurement techniques can be found in refs [19-21]. Moment generated by the fan is an indication of power consumption by the fan. Fan moment was measured in the following way. The friction rig was run at different rpm with fan in the casing. Moment was measured at specific rpm. Moment was measured again at the same specified rpm while decreasing the rpm. This establishes the repeatability of the measured data. After this, the fan alone was removed from the setup and the same measurement method was repeated. The difference between the two measurements (with and without fan) at a particular rpm gives the moment of the fan. In CFD simulation, moment was calculated from the contribution of pressure and shear forces acting on the fan surfaces. Table 2 shows the comparison and method of calculating torque values. Flow data are non-dimensionalized as described in the definitions [22].

3. THE CFD MODEL

3.1. Theoretical background

Interested readers can skip this section and go to results and discussion section. CFD model used in industrial applications has now become standard and main purpose of this section is to provide a theoretical connection between the experiments and numerical simulation. We briefly describe the moving reference frame along the conservation equations that are solved numerically to obtain the turbulent flow field in the rotating fan system. The geometrical model is same as described earlier in the experimental setup. For incompressible flow the governing continuity and momentum equations Reynolds averaged Navier-stokes equations (RANS) has following form,

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{2}$$

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -P \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu \frac{\partial u_i}{\partial x_j} - \overline{u_i^{'} u_j^{'}}\right] \tag{3}$$

where ρ is the density, μ is the dynamic viscosity, U(**U,V,W**) is the velocity in x,y and z direction, p is the pressure, primes denote fluctuating components. The additional term, $u_i u_j$ in the momentum equation. is called Reynolds stress tensor.

The system of equations has more unknown variables than equations to solve and is therefore not closed. The Boussinesq hypothesis relates the Reynolds stress to the mean flow velocity gradients and can be expressed as in equation (4)

$$-\rho \overline{u_i^{'} u_j^{'}} = \mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_i}{\partial x_j} \right) - \frac{2}{3} \rho \delta_{ij} k \tag{4}$$

where δ_{ij} is the kronecker delta and *k* the turbulent kinetic energy, which is defined as

$$k = \frac{1}{2}\overline{u_i' u_i'} \tag{5}$$

$$\mu_t = C_\mu \rho \frac{k^2}{\varepsilon} \tag{6}$$

For the computations performed in this paper, the realizable k-ε turbulence model was used. In the k-ε model the turbulent viscosity (μ_t) was achieved by solving two transport equations, one for the turbulent kinetic energy (*k*) and one for the turbulent dissipation rate (ε) (equations not shown).

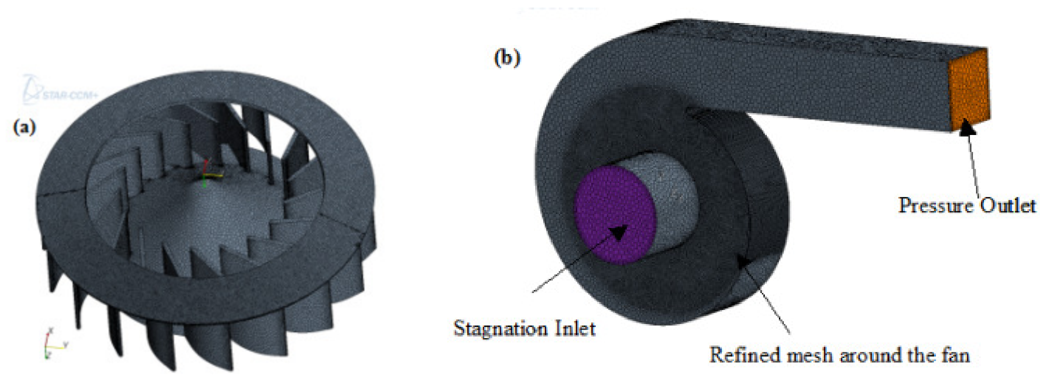

Figure 4. (a) Surface mesh and (b) Volume mesh generated in Star CCM +.

3.2. Moving reference frame model

The steady state approximation in moving reference frame (MRF) model, allows individual cell zones to rotate or translate with different speeds. MRF model is used as fan is the rotating member. This is achieved by dividing the domain into separate zones where the flow is solved in stationary or rotating coordinate systems.

To transform the fluid velocities form stationary to rotating frames,

$$\begin{aligned} \vec{u}_r &= \vec{u} - \vec{v}_r \\ \vec{v}_r &= \vec{\omega} \times \vec{r} \end{aligned} \tag{7}$$

where $\vec{u}_r$ is the velocity relative to the rotating frame, $\vec{u}$ is the absolute velocity and $\vec{v}_r$ is whirl velocity (due to moving frame). $\vec{\omega}$ is angular velocity and $\vec{r}$ is the position vector to the rotating frame.

Solving the equations of motion in the rotating reference frame results in additional terms in the momentum equation,

$$\frac{\partial \rho}{\partial t} + \nabla . \rho \vec{u}_r = 0 \tag{8}$$

$$\frac{\partial}{\partial t}\rho\vec{u} + \nabla.(\rho\vec{u}_r\vec{u}) + \rho(\vec{\omega}\times\vec{u}) = -\nabla p + \nabla\bar{\bar{\tau}} + \vec{F} \quad (9)$$

where $\bar{\bar{\tau}}$ is the viscous stress.

The coriolis and centripetal accelerations are included in the momentum equation with the term ($\vec{\omega}\times\vec{u}$). These equations are solved using commercially available Star CCM+ software.

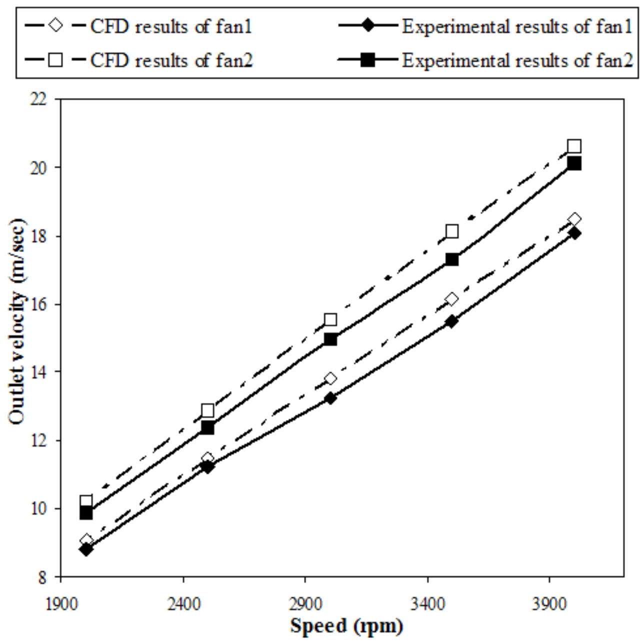

Figure 5. Validation of the numerical model with experimental results.

3.3. Mesh and boundary conditions

The physical configuration and computational domain of the fan and volute is depicted in Fig. 4. In all the simulation the volute casing was the same and only the fan design was changed. The geometry cleaning and surface meshing was carried out in HyperMesh, a commercially available software. Polyhedral volume mesh for the domain was generated in Star CCM+.

The mesh sizes for the fan and volute casing was about 5 lakhs cells. This was arrived after the grid independence study with mesh size varying from 1 to 7 lakh cells. In the rotary fan region, fine mesh was generated and care was taken such that the value of wall y+ does not exceed 15. For validation we choose two fans with 12 (fan 1) and 18 blades (fan 2) and results are compared with experiments. Other details are given in table 1. Stagnation inlet, pressure outlet, and rpm of the fan was given as boundary conditions in order to replicate the physical model (see Fig. 3) in the numerical model. Angle definition is shown in Fig. 1(b). We have used typical values of fan parameters used in the automotive industries.

Table 1. Configuration of fans used in experiments. Dimensions are in mm.

	No. of blades	Inlet angle	Outlet angle	ID of fan	OD of fan	Thickness of blade
Fan 1	12	68	44	90	132	1.6
Fan 2	18	52	44	90	132	1.6

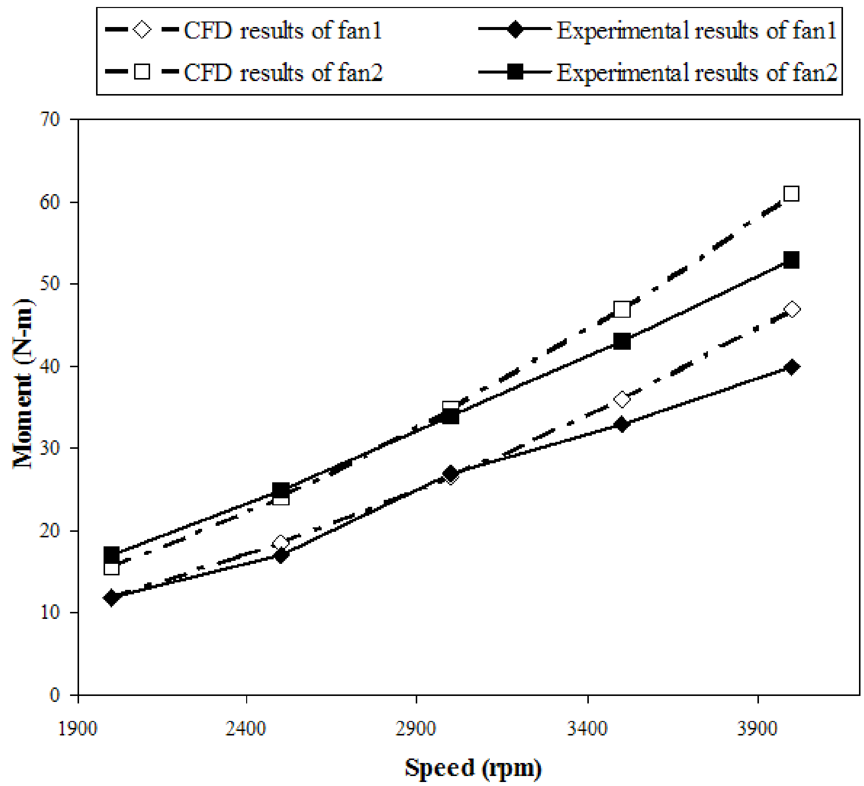

Figure 6. Validation of numerical model for momentum of the fan.

3.4. Validation of the numerical model

Fig. 5 shows the comparison of outlet velocities obtained from experiments and the CFD model. As expected the flow velocity increase with increasing rpm. CFD results are in good agreement with test results for both the fan 1 and fan 2. The maximum error between the CFD model and the physical model is approximately 3%, which is within the acceptable limit. It is to be noticed that error increases as speed increases. Torque (or moment) required to rotate the fan at a given speed is also compared with experimental and CFD results. Table 2 shows the moment of the fan from experiments and CFD simulation. Moment due to fan in an important factor for automotive industries where power consumption due to fan is a great concern. In the competitive market, automobile industries are under pressure to improve vehicles mileage with minimum losses in the engine cooling system. Further, understanding of fan power consumption is important for optimization [23, 24]. As described earlier, moment due to fan was measured by measuring moments with and without fan and then subtracting it. Fig. 6 shows the comparison between the experimental data and CFD results. The results show that fan consumes more power at higher speeds. Two points are noted here: First, results compares well between experiment and CFD at lower speeds. Second, the results show the tendency of divergence at higher speeds. The mismatch of the results at higher speeds may be due to the increase in frictional resistance of the mounting plate and fan, the torsional vibration loads exerted on the shaft at higher rotational speeds.

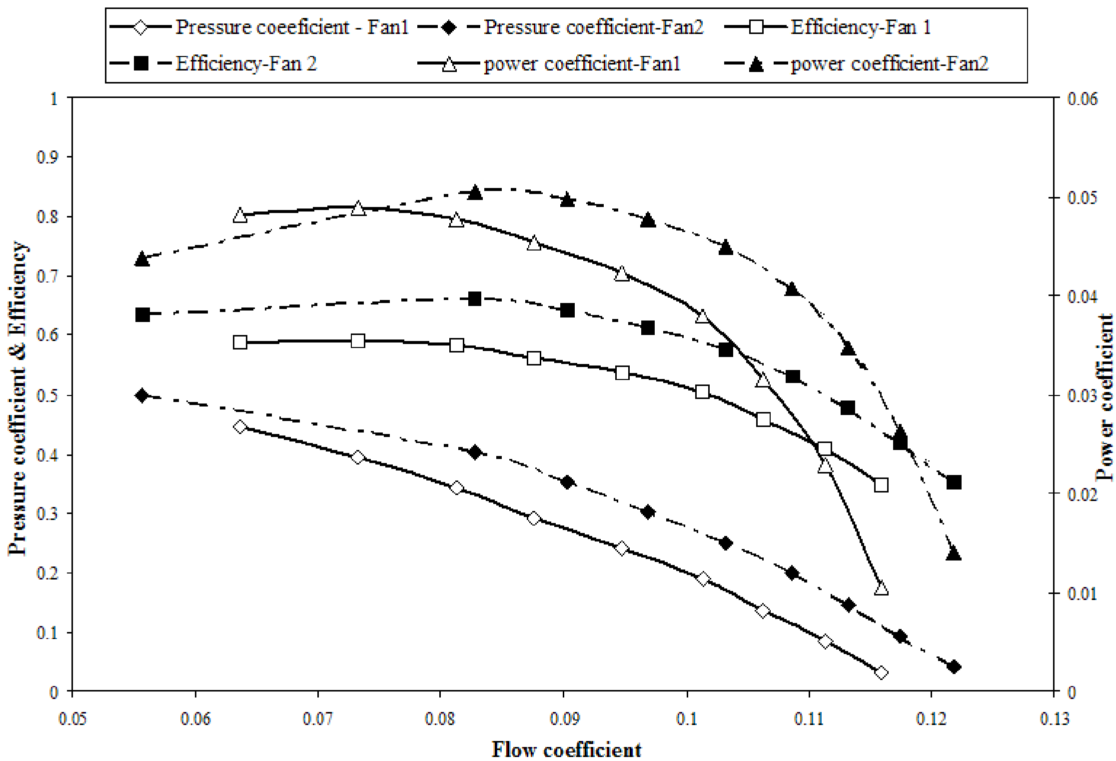

Figure 7. Performance characteristics comparison fan 1 (12 blades) and fan 2 (22 blades) at 4000 rpm.

Table 2. Momentum table for combined fan with mounting plate and mounting plate alone.

Speed (RPM)	Without fans (experiment, Nm×10^3)	With fan 1 (experiment, Nm×10^3)	Fan 1 (experiment, Nm×10^3)	Fan 1 (Simulation, Nm×10^3)	With fan 2 (experiment, Nm×10^3)	Fan 2 (experiment, Nm×10^3)	Fan 2 (Simulation, Nm×10^3)
	(A)	(B)	(A) - (B)		(C)	(A) – (C)	
2000	22	34	12	11.98	39	17	15.60
2500	25	44	17	18.49	50	25	24.19
3000	27	54	27	26.56	61	34	34.75
3500	31	64	33	36.00	74	43	47.00
4000	38	78	40	4700	91	53	61.00

For further simulations we have used the same numerical model for other fan designs. It is to be noticed that close agreement between the simulations and experimental results was possible due to the simplicity of the experimental setup and CFD model.

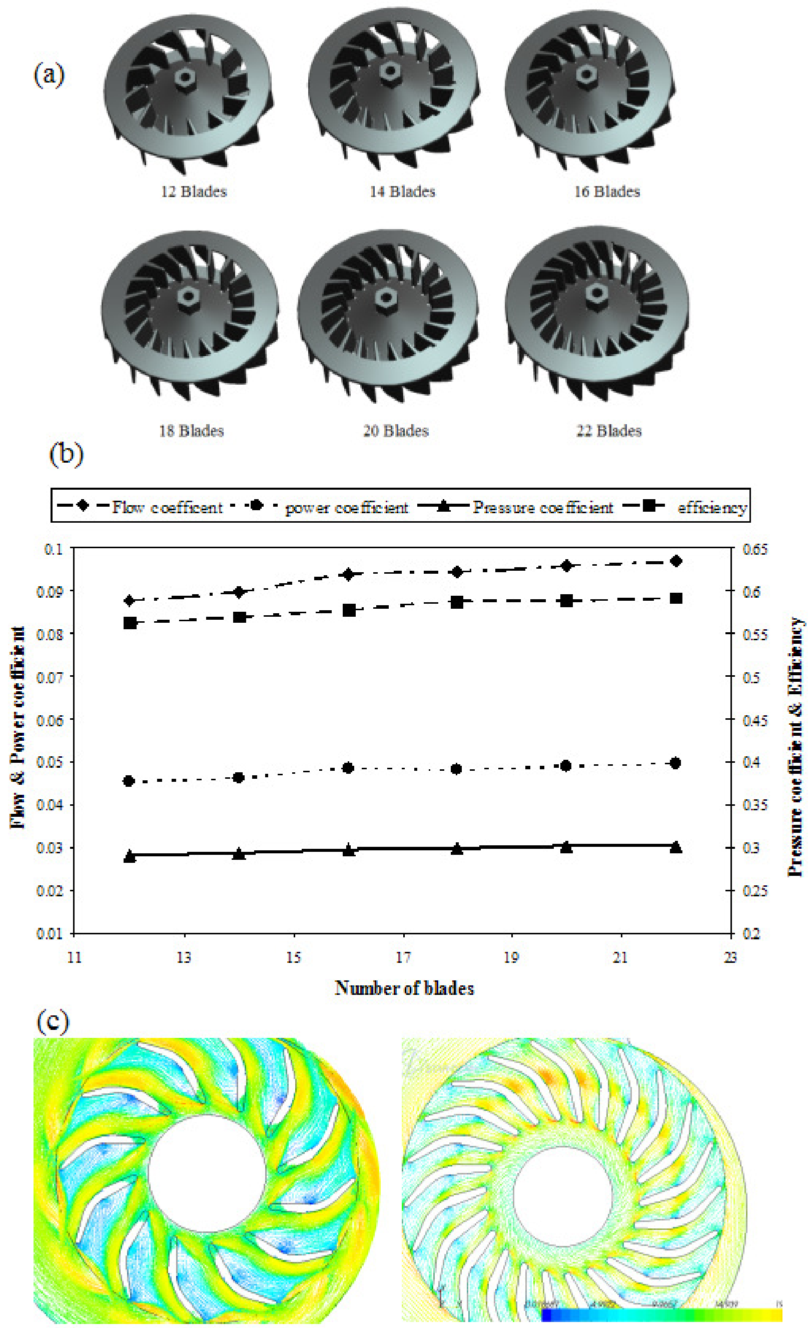

Figure 8. (a) Fan design from 12 to 22 blades, (b) performance characteristics for 12 to 22 blades and (c) relative velocity field for 12 and 22 blades.

4. RESULTS AND DISCUSSION

We pointed out in introduction that fan performance testing should not be done at the vehicle level. We provide a quantitative explanation for this. Fig. 7 shows the variation of pressure coefficient, efficiency and power coefficient as a function of flow coefficient for fan 1 (12 blades) and fan 2 (18 blades). A low flow coefficient represents high resistance system (similar to the actual engine cooling system) and vice-versa. Few interesting observations are made: (a) As the flow coefficient increases the difference between the pressure coefficient of 12 and 18 blades increases and opposite happens when flow coefficient decreases. E.g. at lowest flow coefficient pressure coefficient of fan 2 is only 12% higher than fan 1. However at highest flow coefficient the difference is 30%, (b) Fan efficiency and power coefficient shows similar trends, (c) In the same limit of flow coefficients, efficiency varies from 1.75% to 8.0% and, (d) power coefficient varies between 9% to 34%. To an experimental engineer it means that if they evaluate the fan performance in the actual system they will be able to see only 12%, 1.75% and 9% difference in performance with 12 and 18 blades fan. Hence, a true picture of fan performance may not emerge and fan parameter selection like number of blades would not be optimum. It is important to know the actual flow coefficient as it is used to calculate heat transfer coefficient for engine thermal loads. Data used from the actual engine system can over predict the thermal loads requirements. In the next section we present the effect of various fan parameters on performance.

4.1. Effect of number of blades, N_b

According to Bruno [17] the number of blades in fan cannot be determined theoretically. However, it is time consuming and costly to determine N_b experimentally as it requires large number of prototypes of fan to be made. CFD has become an important tool to investigate such kind of problems. The performance characteristics of the backward curved fans for N_b = 12, 14, 16, 18, 20 and 22 blades is shown in Fig. 8 from the CFD model. Other fan parameters were kept constant. The main purpose this discussion is to provide information about the percentage change in fan performance due to fan blades alone. From 12 blades to 22 blades, the gain in pressure coefficient, efficiency and flow coefficient is 4%, 5% and 10.6% respectively. It is to be noted that from 12 to 22 blades, mass of the fan blades has increases about 80% whereas the performance has not improved proportionally. The improvement in the performance from 12 to 22 blades can be understood from the relative velocity field shown in Fig. 8(c). The relative velocity in the blade passage becomes more uniform due to proper guidance as N_b increases and hence wakes regions decreases. This could reduce noise generated due to wake formation [25-29]. Formation of wake region is one of the major contributors to the fan losses [30,31]. Further increase in N_b would deteriorate the fan performance and boundary layer effects may become dominant.

4.2. Effect of outlet angle

The effect of outlet angle on performance characteristics of fan is presented in Fig. 9. The present study was carried out for outlet angle (Φ_o) of 34°, 44° and 54° to visualize the effect of outlet angle. $\Phi_o = \Phi_o = 90^0$ represents a radial blade fan. One simulation was also performed with forward curve fan with outlet angle of -20^0 in the opposite direction (Fig. 9a (IV)). It is noticed from the figure that wake region increases as outlet angle increases and forward curved blades shows more re-circulation zones. This is undesirable as it may increase wideband sound power level [32, 26, 29]. These re-circulation zones results in transient pressure fluctuation [33-34] resulting noise generation [35-36]. However, flow coefficient is high for forward curved fan (Fig. 9b), which comes at the expense of high power coefficient. From Fig. 9(b), for 21% increase in flow coefficient from Φ_o = 34° to Φ_o = -20°, increase in power coefficient is 42%. On the other hand, efficiency decreased by 4.5%. It means that increase in fan flow rate is accompanied by high power consumption and decrease in fan efficiency.

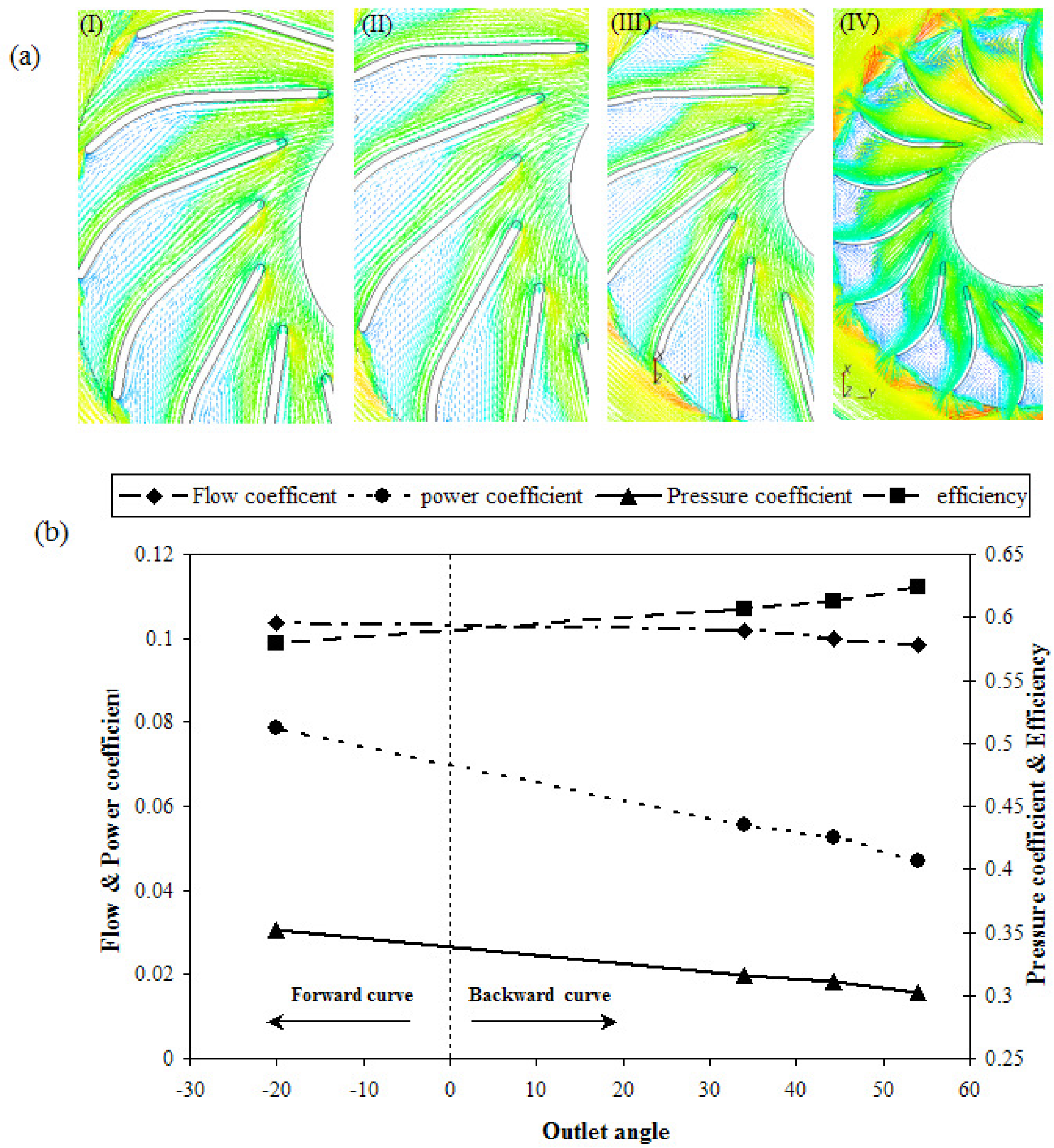

Figure 9. (a) Formation of re-circulation zones for different outlet angles of 34°, 44°, 54° and forward curved blade (-200), (b) fan characteristics at these angles at 4000 rpm.

Increase in power consumption is always a concern for automotive industries as it can reduce engine performance in terms of mileage. This forward and the backward curved fan were tested on a scooter engine at various speeds. It was observed that temperature reduction of the engine surfaces were more with forward curved fan. Further, the vehicle was run on the test track to find out any significant reduction in the vehicle's mileage due to forward and backward fan. Each fan was tested on four different vehicles to mitigate the variability from vehicles to vehicles. It was observed the component durability was higher with forward fan. Further, difference in mileage (in terms of km/liter of petrol consumed) was within 2%, which is less than the error in measurement itself. Hence, it is suggested that forward curved fan should be used wherever temperature related problems occur in the engine especially in high displacement engines without worrying about the high power consumption and its low efficiency. However, increase in flow coefficient could lead to undesirable noise generation [37-40], which should be looked upon before implementation. Several studies have been done [41-47] to control the noise generated

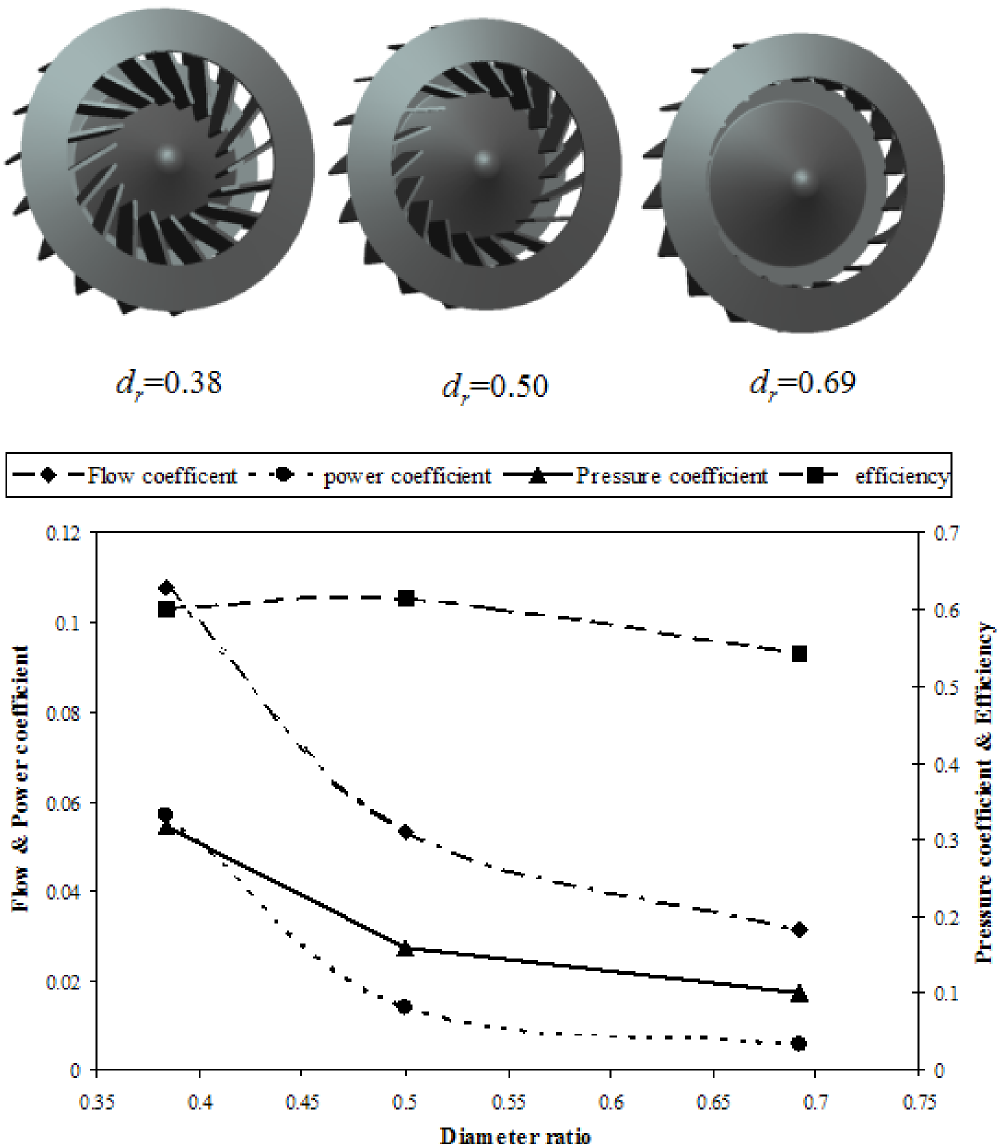

Figure 10. Performance curves for diameter ratios 0.38, 0.5, and 0.69 at 4000 rpm.

by the fan. Other parameters like blade shape [48-50] and inlet angle [51] also affects fan performance, which needs to be investigated further.

4.3. Effect of diameter ratio

The diameter ratio, d_r (ratio of inner blade tip length by outer blade tip length from the center) is yet another important parameter that affects the fan performance. Larger the d_r smaller is the blade length (see Fig. 10a). The present analysis was carried out for different d_r, by varying inner diameter and keeping outer diameter constant. Fig. 10(b) shows the effect of three d_r (0.38, 0.50 and 0. 69). The efficiency of the fan increases and then decreases as d_r increases. Highest efficiency was observed for d_r = 0.50 even though flow, pressure and power coefficient shows a decreasing trend.

5. CONCLUSIONS

In this paper, investigations on the effect of centrifugal fan parameters on performance has been presented thorough experiments and CFD simulations has been presented. Test results show that forward curved fan with higher mass flow rates has negligible effect on the vehicle's mileage (fuel consumption) when compared with backward fan with lower flow rates. Following are major conclusions we draw from the present investigations:

1. Increase in the number of blades increases the flow coefficient accompanied by increase in power coefficient. However, difference in the performance (efficiency, flow and power coefficient) tends to decrease at higher pressure coefficient. Hence, it is concluded that fan performance measurement under high pressure coefficient would not provide the true measure of the fan characteristics. Under high pressure coefficient all fan behave similarly.
2. Increase in the number of blades increases the flow coefficient and efficiency due to better flow guidance and reduced losses.
3. In the range of parameters considered, forward curved blades have 4.5% lower efficiency with 21% higher mass flow rates and 42% higher power consumption compared to backward curved fan. Experimental investigations suggest that engine temperature drop is significant with forward curved blade fan with insignificant effect on mileage. Hence, use of forward fan is recommended on the vehicles where cooling requirements are high.
4. The efficiency of the fan first increases and then decreases with diameter ratio. The best efficiency of the fan was observed to be at diameter ratio of 0.5.

REFERENCES

[1] Jeon, Wan-Ho; Baek, Seung-Jo; Kim, Chang-Joon, Analysis of the aeroacoustic characteristics of the centrifugal fan in a vacuum cleaner, Journal of sound and vibration, Vol. 268, Issue 5, pp. 1025-1035, 2003

[2] Hongmin Li, Flow driven by a stamped metal cooling fan – Numerical model and validation, Experimental thermal and fluid science, Vol. 33, Issue 4, pp. 683-694, 2009.

[3] Yan Jianhua,Tang Weixin, Yuan Wenhua, Optimizing technique of air-cooled engine centrifugal cooking fan test, Chinese internal combustion engine engineering, DOI: cnki:ISSN:1002-6819.0.2001-06-006, 2002-01.

[4] Chen Rugang et al., Experimental study on erosion characteristic for different blade contours in the centrifugal fan exposed to solid particulate environment, Fluid mechinary, DOI: cnki:ISSN:34-1071.0.2000-01-000, 2000-01.

[5] Chih-Chung Chang,1 Chiao-Hung Cheng,1 Ming-Tsun Ke,2 and Sih-Li Chen1, Experimental and numerical investigations of air cooling for a large-scale motor, International Journal of Rotating Machinery, doi:10.1155/2009/612723, Vol. 2009, 2009.

[6] Woodard Jr. J. K., Johnson G. E., Optimal design of cooling fan for industrial electric motors, Journal of mechanisms, transmissions, and automation in design, 108 (2), pp. 224-266, 1986.

[7] Sheam-Chyun Lin, and Chia-Lieh Huang, An integrated experimental and numerical study of forward–curved centrifugal fan, experimental thermal and fluid science, Vol. 26, Issue 5, PP. 421-434, 2002.

[8] Baranski, B. R., Designing the engine cooling fan, Technical paper series, No. 740691, pp. 1-8, 1974.

[9] Smith, J. D., Reynolds J. R., Engine cooling fan and shroud optimization for blow through operation, SAE technical paper series, No. 860764, pp. 1-6, 1986.

[10] Campbell, K., Engine cooling fan theory and practice, Society of automotive engineers – Journal (The SAE journal), 52 (9), pp. 408-420, 1944.

[11] Kiura, T., Kushibiki T.,Shimoi T., Research on prediction of cooling air flow in small air-cooling utility engines, SAE technical paper # 2005-32-0023/JSAE 20056606, 2005.

[12] Coggiola E., Dessale B., CFD based Designs for automotive engine cooling fan systems, SAE technical paper # 980427, 1998.

[13] Vishal K., O. P. Singh, Manish G,. Yatin, V. C., Optimization of Air-Cooling System of 4-Stroke Scooter Engine, SAE technical paper # 2008-32-0071 (SAE) / 20084771 (JSAE), 2008.

[14] Manish G., Kumar O. H., Prasad N., Swamy V., Experimental and CFD simulation-based analytical optimization of air-cooling system for a small 4-stroke engine, SAE technical Paper #2005-32-0026/JSAE 20056607, 2005.

[15] F. S. Howes and R. R. Real, Noise origin, power, and spectra of ducted centrifugal fans, J. Acoust. Soc. Am. Vol. 30, Issue 8, pp. 714-720, 1958.

[16] Frank P. Bleier, Fan handbook: selection, application and design, Mcgraw-hill Professional Publishing, 1997.

[17] Bruno E., Design and operation of centrifugal, axial-flow and cross flow fans, Pergamon press, 1962.

[18] Sreenivasulu T., Siva Prasad G., Vishal K., Manish G., Pattabhiraman V., Design and Optimisation of Cooling Fan for Forced Air-cooled Engines, STAR Indian Conference, Chennai, 2009.

[19] American national standard ANSI/ASHRAE 51-1987, Laboratory methods of testing fans for rating, 1987

[20] Shepherd, I. C., A four hole pressure probe for fluid flow measurements in three-dimensions, J. Fluids Eng., Trans. ASME, 103, 590-594, 1981.

[21] British standards BS 1042: section 1.1: 1981, Methods of measurement of fluid flow in closed conduits, BSI, London, 1981.

[22] Xiaomin L., Qun D., Guang X., Engineering application of computational fluid mechanics, vol. 2 (2) 130-140, 2008.

[23] Wang, D. G, Muller, P. K., Improving cooling efficiency by increasing fan power usage, Microelectronics Journal 31, 765–771, 2000.

[24] F. S. Howes and R. R. Real, Noise origin, power, and spectra of ducted centrifugal fans J. Acoust. Soc. Am. Vol. 30, Issue 8, pp. 714-720, 1958.

[25] Edmane Envia, Fan noise reduction: an overview, International Journal of Aeroacoustics, Vol.1(1), 2002.

[26] Thomas Bouquet, Farid Bakir,Smaïne Kouidri, Robert Rey, Study of the 3D flows in the forward-curved blades centrifugal fans, ASME Joint U.S.-European Fluids Engineering Division Conference (FEDSM2002), Montreal, Quebec, Canada, Paper no. FEDSM2002-31102 pp. 1019-1025, doi: 10.1115/FEDSM2002-31102, 2002.

[27] Nashimoto, A., Measurement of aerodynamics noise and wake flow field in cooling fan with winglets, Journal of visualization, 7(1), pp. 85-92, 2004.

[28] Vladimir G. Sapojnikoff, Tips of getting the most cooling from the right cooling fan, Electromechanical design, 16 (10), pp. 18-21, 1972.

[29] Beier K. A., Weir T. J., Methods to reduce cooling fan noise, Noise control: A publication of the acoustical society of America, 56C, pp. 1-9, 1958.

[30] Rafael Ballesteros-Tajadura, Sandra Velarde-Suárez, Juan Pablo Hurtado-Cruz, and Carlos Santolaria-Morros , Numerical calculation of pressure fluctuations in the volute of a centrifugal fan, J. Fluids Eng., Vol. 128, Issue 2, 359, doi:10.1115/1.2170121, 2006.

[31] M. Younsi, F. Bakir, S. Kouidri, R. Rey, Influence of design parameters on the unsteady flow in a centrifugal fan, ASME/JSME 5th Joint Fluids Engineering Conference (FEDSM2007), San Diego, California, USA, Paper no. FEDSM2007-37609, pp.1011-1022, doi: 10.1115/FEDSM2007-37609, 2007.

[32] Bommes, Leonhard, Grundmann, Reinhard, Klaes, Klaus, Kramer, Carl, Effects of blade design on centrifugal fan noise and performance, Noise Control Engineering Journal. Vol. 43(4), pp. 91-101, 1995.

[33] Tarek Meakhail and Seung O. Park, A study of impeller-diffuser-volute interaction in a centrifugal fan, J. Turbomach., Vol. 127, Issue 1, 84, doi:10.1115/1.1812318, 2005.

[34] Younsi M., Bakir F., Kouidri S., and Rey R., Influence of impeller geometry on the unsteady flow in a centrifugal fan: numerical and experimental analyses, International Journal of Rotating Machinery, Vol. 2007, Article ID 34901, doi:10.1155/2007/34901, 2007.

[35] Rafael Ballesteros-Tajadura, Sandra Velarde-Suárez, Juan Pablo Hurtado-Cruz, and Carlos Santolaria-Morros , Numerical calculation of pressure fluctuations in the volute of a centrifugal fan, J. Fluids Eng., Vol. 128, Issue 2, 359, doi:10.1115/1.2170121, 2006.

[36] M. Younsi, F. Bakir, S. Kouidri, R. Rey, Influence of design parameters on the unsteady flow in a centrifugal fan, ASME/JSME 5th Joint Fluids Engineering Conference (FEDSM2007), San Diego, California, USA, Paper no. FEDSM2007-37609, pp.1011-1022, doi: 10.1115/FEDSM2007-37609, 2007.

[37] Morinushi, K., J. The influence of geometric parameters on F.C. centrifugal fan noise. vibrat. acoust. stress reliab. des. Vol. 109, (3), pp. 227-234. 1987.

[38] Q Liu, D Qi, Y Mao, Numerical calculation of centrifugal fan noise, Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, Vol 220 (8) pp.1167-1177, 2006.

[39] W. Neise, Noise reduction in centrifugal fans: A literature survey, Journal of Sound and Vibration, Vol 45 (3), pp. 375-403, 1976.

[40] Jeon, Wan-Ho, A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan, Journal of Sound and Vibration, Vol 265 (1), pp. 221-230. 2003.

[41] Morinushi, K., The influence of geometric parameters on F.C. centrifugal fan noise. J. vibrat. acoust. stress reliab. des. Vol. 109, no. 3, pp. 227-234. 1987.

[42] Sandra Velarde-Suárez, Rafael Ballesteros-Tajadura, Juan Pablo Hurtado-Cruz and Carlos Santolaria-Morros, Experimental determination of the tonal noise sources in a centrifugal fan, Journal of sound and vibration, Vol. 295, Issues 3-5, pp. 781-796, 2006.

[43] Yu Shenbo et al., Design of low noise centrifugal fan, Fluid machinery, DOI: cnki: ISSN: 1005-0329.0.1999-11-003, 1999-11.

[44] Neise W. and Koopmann G. H., Reduction of centrifugal fan noise by use of resonators

[45] Journal of sound and vibration, Vol. 73, Issue 2, pp. 297-308, 1980.

[46] Sandra Velarde-Suárez, Carlos Santolaria-Morros, and Rafael Ballesteros-Tajadura, Experimental study on the aeroacoustic behavior of a forward-curved blades centrifugal fan, J. Fluids Eng.,Vol.121, Issue2, 276, doi:10.1115/1.2822202, 1999.

[47] Esra Sorguven, Yilmaz Dogan, Faruk Bayraktar, Ergin Arslan, Computational Noise Prediction of a Centrifugal Fan, ASME International Mechanical Engineering Congress and Exposition (IMECE2007), Paper no. IMECE2007-41842 pp. 1427-1432, doi: 10.1115/IMECE2007-41842, Seattle, Washington, USA, 2007.

[48] Kwang-Yong Kim and Seoung-Jin Seo, Shape optimization of forward-curved-blade centrifugal fan with navier-stokes analysis, J. Fluids Eng. -- Vol. 126, Issue 5, 735, doi:10.1115/1.1792256 , 2004

[49] Kwang-Yong KIM and Seoung-Jin SEO, Application of numerical optimization technique to design of forward-curved blades centrifugal fan, JSME International Journal Series B, Vol. 49, No. 1, Special Issue on Urban Air and Environment Control Technologies pp.152-158, 2006.

[50] Tsutomu Adachi, Naohiro Sugita, and Yousuke Yamada, Study on the performance of a sirocco fan (optimum design of blade shape), International Journal of Rotating Machinery, Vol. 7 Issue 6, Pages 405-414, doi:10.1155/S1023621X01000355, 2001.

[51] Yu, Z., Li S., He W., Numerical simulation of flow field for whole centrifugal fan and analysis of the effects of blade inlet angle and impeller gap, HVAC and R research, 11(2), pp. 263-283, 2005.

Author's biography

Dr. Om Prakash Singh is working in TVS Motor Company in the R&D department since March 2006. He is responsible for thermal management of internal combustion engines. His research interest are multidisciplinary in nature in the fields of heat and mass transfer, NVH (noise, vibration and harshness), engine tribology, failure analysis, CFD (computational fluid dynamics), FEA (finite element analysis) and other related fields. Prior to joining the company, he finished his PhD in mechanical engineering from Indian Institute of Science, Bangalore, India in 2006 in the area of heat and mass transfer and double-diffusive convection. His detailed profile can be seen on his website: www.omprakashsingh.com.

Rakesh Khilwani is a Doctoral Student at the Multiscale Manufacturing and Dynamics Laboratory at Carnegie Mellon University. He obtained his Bachelors of Technology and Masters of Technology from Indian Institute of Technology Madras. He majored in Mechanical Engineering and specialized in Intelligent Manufacturing. The work in this paper was done during his summer internship program at TVS Motor Company.

T. Sreenivasulu is a CFD & thermal system analyst interested in the development of new concepts and paradigms in the area of thermal system applications for various domains like automobile, power plant. He is a post graduate in thermal engineering from Indian Institute of Technology Madras, Chennai. After post graduation he joined TVS Motors Company, R&D in Hosur, India. He had published two international journals, 8 international & national conferences. He has filed 10 patents.