J. Inst. Eng. India Ser. C
DOI 10.1007/s40032-017-0356-z

ORIGINAL CONTRIBUTION

Design, Development and Analysis of Centrifugal Blower

Beena Devendra Baloni[1] · Salim Abbasbhai Channiwala[1] ·
Sugnanam Naga Ramannath Harsha[2]

Received: 12 June 2015 / Accepted: 24 March 2017

Abstract Centrifugal blowers are widely used turbomachines equipment in all kinds of modern and domestic life. Manufacturing of blowers seldom follow an optimum design solution for individual blower. Although centrifugal blowers are developed as highly efficient machines, design is still based on various empirical and semi empirical rules proposed by fan designers. There are different methodologies used to design the impeller and other components of blowers. The objective of present study is to study explicit design methodologies and tracing unified design to get better design point performance. This unified design methodology is based more on fundamental concepts and minimum assumptions. Parametric study is also carried out for the effect of design parameters on pressure ratio and their interdependency in the design. The code is developed based on a unified design using C programming. Numerical analysis is carried out to check the flow parameters inside the blower. Two blowers, one based on the present design and other on industrial design, are developed with a standard OEM blower manufacturing unit. A comparison of both designs is done based on experimental performance analysis as per IS standard. The results suggest better efficiency and more flow rate for the same pressure head in case of the present design compared with industrial one.

✉ Beena Devendra Baloni
beena_baloni@yahoo.co.in; pbr@med.svnit.ac.in

[1] Mechanical Engineering Department, Sardar Vallabhbhai National Institute of Technology, Ichchhanath, Surat 395007, Gujarat, India

[2] Innominds Software SEZ India Pvt Limited, Waverock, Nanakramguda 500008, Hyderabad, India

Keywords Centrifugal blower · Unified design · Numerical analysis · Performance analysis

Notations

b	Width
d	Diameter
g	Gravitational acceleration
H	Head
k	Constant
N	Speed of rotation
N_s	Specific speed
NT	New temperature
Q	Flow Rate
r	Radius
T	Temperature
u	Blade velocity
v	Absolute velocity
v_w	Whirl velocity
W	Relative velocity
z	Number of blades
β	Blade angle
Δp	Pressure rise
ε	Pressure ratio
η	Efficiency
ρ	Fluid density
σ	Speed coefficient
σ_c	Solidity
Φ	Volume coefficient
Ψ	Pressure coefficient

Subscripts

1	Impeller entry
2	Impeller outlet
eff	Effective
vir	Virtual

Introduction

Now-a-days, the world is facing serious challenges in energy. Industries consume almost 70% of total energy generated. One of the biggest electricity consumers, in the industrial sector, is the motor system. In case of the power consuming machine, centrifugal machines dominate among total fan and blower market by consuming 90% of sector energy [1]. Usually blowers are purchased solely based on their cost. So, little attention is given to monitor and evaluate their reliability, economic life span and their optimum efficiency. Therefore, there is a requirement of enhancement in the design of centrifugal blowers, to increase their performance.

Primary attempt to compile the design methodology for pumps and blowers has been made by Church [2]. Eck [3], Osborne [4], Whitfield and Banes [5] had extended the work of the church and presented the details for fans and blowers. Yan, et al. [6] have investigated internal relations between a large number aerodynamic sketches and performance parameters of centrifugal fans by statistical data by introducing the two important parameters, specific speed (Ns) and specific diameter (Ds) and their relation with flow coefficient and pressure coefficient. Yan and Xi [7] investigated the traditional design methods and suggests that, corresponding relations exist not only between specific speed and specific diameter but also with specific speed and exit width of the blade. This leads to deduction of a relation between an inlet and outlet diameter ratio and the flow coefficient, in response to the principle that the minimum loss occur in the fluid passages of impeller. Mishra [8] also studied and made an attempt to compile design methodology, for the radial tipped centrifugal blower. An experimental comparative assessment of design methodologies, suggested by the church, Osborne and methods laid down from fundamental principles of fluid flow and energy is done by Meakhail and Park [9]. Masutage, et al. [10] performed simulation and numerical analysis of the unified design of the fan, and concluded that with the increase in number of blades, the circulatory flow reduced in the blade passage and more energized flow was developed. The preliminary design of the volute is based on the assumption of an adiabatic one dimensional incompressible flow [11]. Design methods for volutes are usually based either on the assumption of constant average velocity (Stepanoff, 1948) or on the assumption of constant angular momentum (Pfleiderer, 1949). Baloni, et al. [12] had made an experimental study of a centrifugal blower and results suggested better flow conditions in volute of 'constant mean velocity' design concept compare to 'constant angular momentum (CAM)'. However, the proper design of volute should give the uniform distribution of pressure at a volute inlet or impeller outlet periphery [13–16]. The outlet blade angle β_2, diameter ratio of impeller inlet and outlet, the outlet blade width and the number of blades were considered as the important geometrical parameters in designing a turbomachine as they were responsible for the variation of head, power etc [2, 17–19]. The optimum number of blades of a radial impeller can only be truly ascertained by experiments. Thus, the blade number is considered as an important parameter in designing of the blower [2, 9, 20–22]. Also, the slip factor has a significant effect on centrifugal blower design and performance. Empirical correlations used to estimate the slip factor provided a constant for a given impeller only at the best efficiency point [5, 23, 24].

The literature survey revealed that much research work is done towards more and more systematic design approach for centrifugal blower/fan. Different authors have suggested different procedures, and each has used differing concepts. A comprehensive theoretical method for design of fans is quite difficult due to the complex and 3D nature of the flow within the fan passages. For this reason, designer has to make use of theoretical results as well as empirical rules. Authors try to design the blowers with such theoretical and empirical rules which enhance the performance of a machine with a minimum use of energy. The suggested design procedure is based more on fundamental concepts and minimum assumptions. The validation of design is done with numerical and experimental analysis.

Parametric Study

Literature studies revealed that there are certain parameters which are very crucial in the role of efficiency of the machine and its losses. The study of such parameters and their interdependency is necessary in blower design are discussed as shown in Fig. 1.

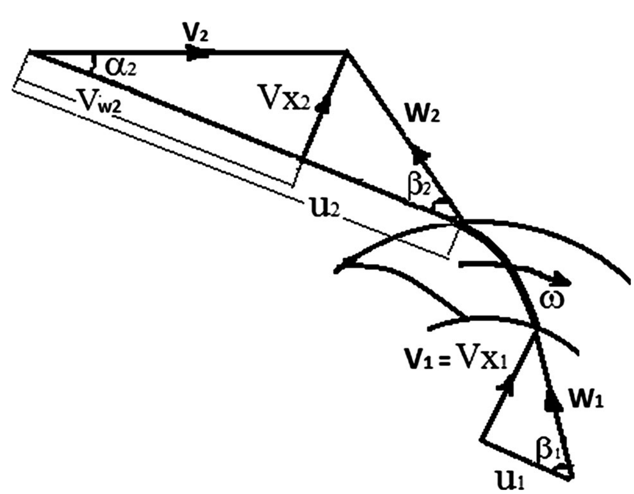

Fig. 1 Inlet and outlet velocity triangle of a backward impeller

Diameter Ratio of Impeller

The impeller losses are dependent on the ratio d_1/d_2 because, the losses expected as the maximum velocity blade encounters is equal to the relative inlet velocity (w_1). Therefore, attention is required in its magnitude. The minimum value further created a relation between d_1/d_2 and φ as shown in Eq. (1) [3].

$$\frac{d_1}{d_2} = 1.194\varphi^{\frac{1}{3}} \tag{1}$$

Inlet Blade Angle

Figure 1 represents the inlet and outlet velocity triangle for better understanding of different velocities and angles at an inlet and outlet of impeller. The relative inlet velocity is a function of an inlet blade angle. Certain experiments were conducted considering the d_1/d_2 and β_1, for the minimum value for d_1/d_2 and the maximum value of β_1. The practical results suggested that the value of β_1 should not exceed 35.26° [3].

Outlet Blade Angle

The outlet blade angle is the important parameter to be accessed. The v_{w2} is dependent on β_2 and thus, the added head is a function of the β_2. If the value of β_2 increases, the value of cot β_2 decreases and hence v_{w2} increases. This increase the head developed by machine.

Blade Number

The optimum number of blades which gives the best efficiency can be chosen by experience for a particular requirement. Increase in a number of blades increases flow coefficient and efficiency, due to better guidance, which reduces losses. Too few blades are unable to fully impose their geometry on the flow whereas; too many of them restrict the flow passage and leads to higher losses. Generally, the number of blades empirical relation is selected depending upon the pitch to chord ratio or solidity.

Solidity

Solidity is defined as the ratio of the distance between the leading and trailing edges, that is, the chord length to pitches, that is, the angular spacing between adjacent leading edges. The solidity of centrifugal cascade σ_c is given as Eq. (2)

$$\sigma_c = \frac{\left(1 - \frac{d_1}{d_2}\right) * Z}{2\Pi \sin \beta_m}, \text{ where } \beta_m = \frac{\beta_1 + \beta_2}{2} \tag{2}$$

The analytical results show that the solidity for centrifugal fans should be as per Eq. (3) [25].

$$\sigma_c \geq 1.48 \tag{3}$$

Design Methodology

In the present design, the weight flow of air per unit of time passing at a point in a blower/fan is considered constant for steady flow. The volume flow will not be constant since the specific weight varies with changes in temperature and pressure of air. The dimensions of the air passage must be calculated in accordance with this variation in volume flow. Input parameters such as Q, Δp, N, p_1 and T_1 are considered. From characteristic gas equations and compressibility effect, average density is calculated as per Eq. (4).

$$\rho = \frac{\rho_1 + \rho_2}{2} \tag{4}$$

The head rise is calculated as per Eq. (5)

$$H = \frac{\Delta p}{\rho g} \tag{5}$$

The non-dimensional parameters are having considerable assistance to manufacturers and users of the fans/blowers. The non-dimensional parameters shown in Eqs. (6) and (7) are calculated as per given design inputs.

$$N_s = N\sqrt{Q} \Big/ \frac{\Delta p^{\frac{3}{4}}}{\rho} \tag{6}$$

$$\sigma = \frac{0.379N\sqrt{Q}}{H^{\frac{3}{4}}} \tag{7}$$

Diameter coefficient (δ) can be found out from Cordier diagram ($\delta \rightarrow \sigma$) [3] for the given value of σ. The graphical form of Cordier diagram is converted into mathematical form by weighted residual method, and a resultant equation of δ for different stages is obtained as per Table 1 [26].

From σ and δ, pressure coefficient and volume coefficient are calculated as per Eqs. (8) and (9), respectively.

$$\psi = \frac{1}{\sigma^2 \delta^2} \tag{8}$$

$$\varphi = \frac{1}{\sigma \delta^3} \tag{9}$$

The relative inlet velocity should minimum inorder to avoid incidence losses. From literature, the minimum

Table 1 The mathematical form of Cordier diagram

Diameter coefficient, $\delta = a\sigma^{-b}$ (where, a and b = constants)	
a	0.99, for $0.1 < \sigma < 0.4$
a	1.5, for $0.4 < \sigma < 2$
b	0.995, for $0.1 < \sigma < 0.4$
b	0.5866, for $0.4 < \sigma < 1$
b	0.505, for $1 < \sigma < 2$

possible relative velocity is found to be a function of β_1 and minimum value deduction led to the relation as per Eq. (1). The d_1 is derived from Eq. (1), whereas, d_2 is obtained with the help of pressure coefficient. The calculation of inlet velocity is done by assuming that the inlet flow velocity is some fractional part of the outlet blade velocity [4] as per Eq. (10).

$$v_1 = ku_2 \tag{10}$$

With the inlet velocity, the β_1 and w_1 are calculated. The β_1 should be less than 35.26° to avoid incidence losses [3]. The calculation of the outlet whirl velocity is done using Euler head. The virtual head, generated from these derived velocities, is calculated by Eq. (11).

$$H_{vir} = \frac{\left(u_2^2 - u_1^2\right) + \left(w_2^2 - w_1^2\right) + \left(v_2^2 - v_1^2\right)}{2g} \tag{11}$$

It may be assumed that owing to the circulatory flow, friction and turbulence in the impeller, 15% of this virtual head is lost. Thus, the effective head is calculated as shown in Eq. (12)

$$H_{eff} = 0.85\, H_{vir} \tag{12}$$

Since, impeller outlet pressure depends upon the effective head so,

$$\varepsilon_{imp}^{0.286} - 1 = \frac{0.286\, g\, H_{eff}}{RT_1} \tag{13}$$

Now, at a new temperature at an outlet (NT_2), ρ_2, Q_{out} and v_1 are calculated. The outlet width of impeller is calculated from the mass flow rate. If the new velocity v_{1n} derived from effective head concept differs with the assumed velocity within the tolerance of 5% then this value of v_1 is selected for further calculations. Otherwise, the k value [Eq. (11)] is increased and the process is repeated until the difference comes within the tolerance limit. The number of blades is calculated by given relation deduced by Eck [3] as Eq. (14)

$$Z = \frac{8.5 \sin\beta_2}{1 - \frac{d_1}{d_2}} \tag{14}$$

Wiesner-correlation is used for an estimation of a slip factor (μ).The inlet duct diameter (d_0) is calculated according to the fundamental design approach, and is given by the relation as per Eq. (15)

$$d_o = 1.1 d_1 \tag{15}$$

The program based on present impeller design methodology is developed with computer language C++ and flow chart for the same is represented in Fig. 2. The volute is designed based on the modified free vortex theory as it gives better results compare to other design of volute [12]. To overcome the non-uniformity observed in CAM [11] and smooth the pressure irregularities around the periphery of impeller [13–15, 27], radial velocity is considered to be constant, that is, $V_1 = Vx_1 = Vx_2$ in modified free vortex method.

For free vortex flow,

$$rv_u = r_1 v_{u1} = r_2 v_{u2} \tag{16}$$

Volume flow rate,

$$Q = \pi D_1 b_1 V_{x1} \tag{17}$$

The radius at a different azimuth angle and tongue angle are calculated as per Eq. (18) [19] and Eq. (19), respectively,

$$r_\theta = r_3 \exp\left(\frac{\theta}{360^0} \frac{Q}{Cb_3}\right) \tag{18}$$

$$\tan\alpha_v = \frac{Vx_2}{Vu_2'} \tag{19}$$

Numerical and Experimental Analysis

The single stage centrifugal blower comprises mainly impeller and volute as shown in Fig. 3 is designed, with the present unified method, for $\Delta p = 3$ kPa, $Q = 2.77$ m^3/s and $N = 2920$ rpm. The details of design dimensions are represented in the Table 2. To check the performance of blower based on suggested unified design, numerical simulation is carried out with a validated case by CFD software GAMBIT and ANSYS-FLUENT [14, 28].

In order to understand the perfect flow interaction, the impeller is meshed with an unstructured grid and the volute is made structured. The governing equations are analysed using a finite volume formulation. The impeller region is operating in a rotating frame and casing region in a stationary frame of reference. Rotor–stator interaction is taken care by mixing plane reference. The velocity inlet and pressure out let boundary conditions are given as an inlet and outlet of fluid domain respectively. The convergence criterion is selected to be e^{-4} and iterations are carried out till convergence. The simulation results suggest that the maximum pressure rise occurred in the impeller itself whereas; a little bit of transformation of energy occurred in the volute. A combination of the head rise at impeller and volute gives the design head of 255.52 m of air out of which the impeller constitutes the head rise of 84.41%. Static pressure is calculated by an area average technique at different angular locations to check the pressure variations around the impeller periphery. Ten equally distributed, angular locations are selected and the variation of static pressure is plotted as shown in Fig. 4. Maximum pressure variation along the radial periphery of an impeller outlet is 9.76%. Thus, numerical analysis of present design

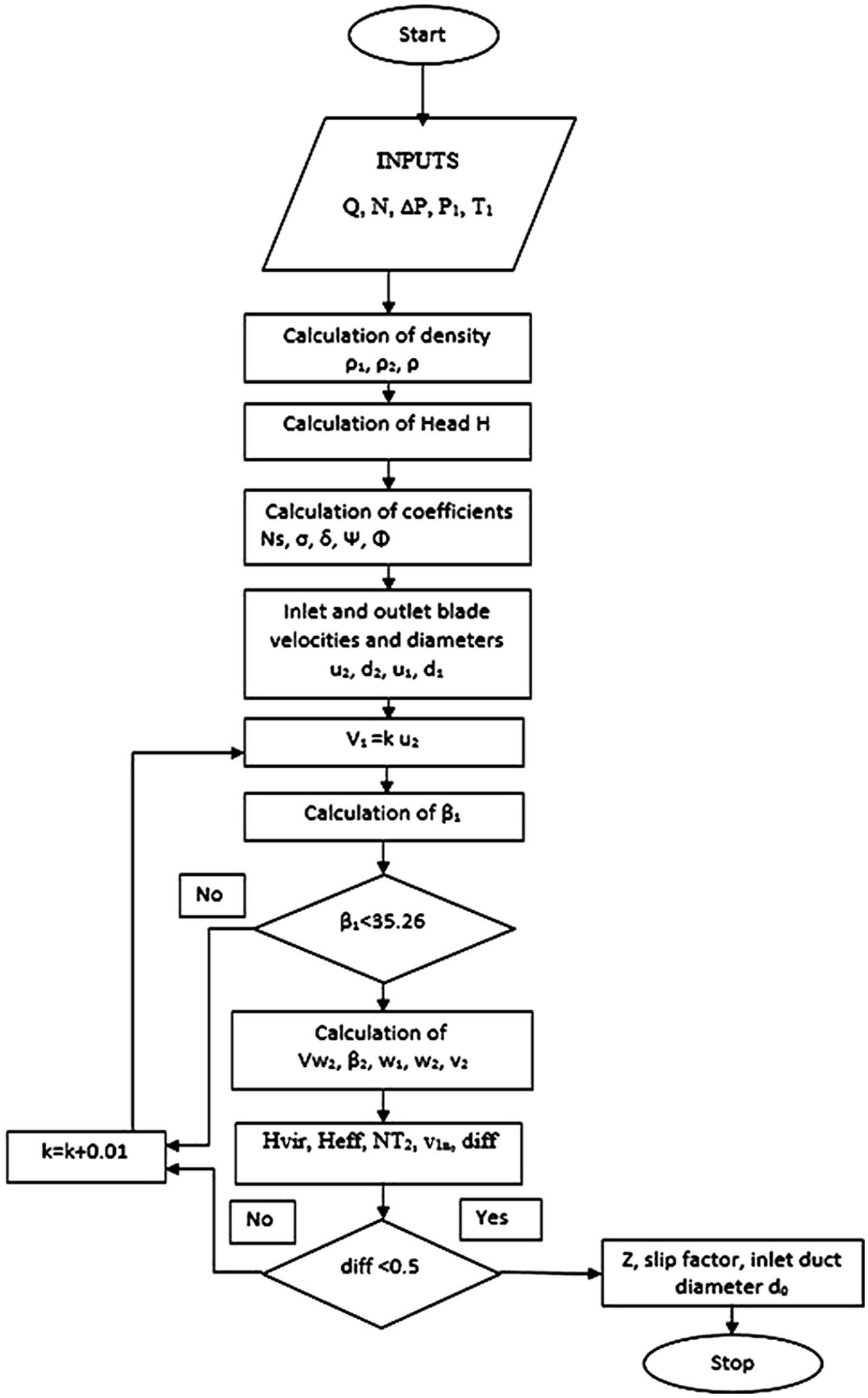

Fig. 2 Flow chart of design of impeller

suggests less variation in static pressure at an impeller outlet which reduces the non-uniformity of flow and enhances the performance of turbomachines [12–15].

After satisfactory results of numerical analysis, present designed blower performance is compared with an industrial OEM supplier blower by experimental analysis. Blowers are manufactured with $Q = 0.5$ m^3/s, $\Delta p = 2.5$ kPa and $N = 2850$ rpm as shown in Fig. 5. The dimensional details of designs are incorporated in the Table 3.

The performance analysis is carried out as per IS: 4894-1987 as shown in Fig. 6. The readings are taken at the same pressure rise conditions for both the blowers and results are plotted as shown in Figs. 7, 8 and 9. Figure 7

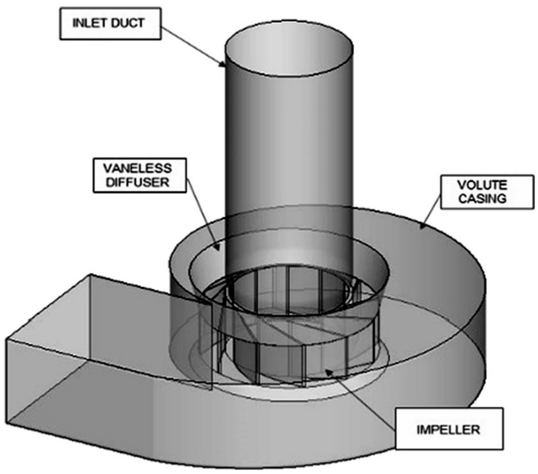

Fig. 3 3D model of single stage centrifugal blower

Table 2 Design parameters of centrifugal blower

Impeller specification		Volute specification	
Parameter	Dimension	Parameter	Dimension
d_1, m	0.34	b_{volute}, m	0.34
d_2, m	0.513	r_3, m	0.282
β_1, degree	16.8°	L_{throat}, m	0.494
β_2, degree	18.4°	α_v, degree	23.7
$b_{impeller}$, m	0.167	r_v, m	0.295
Z	10		

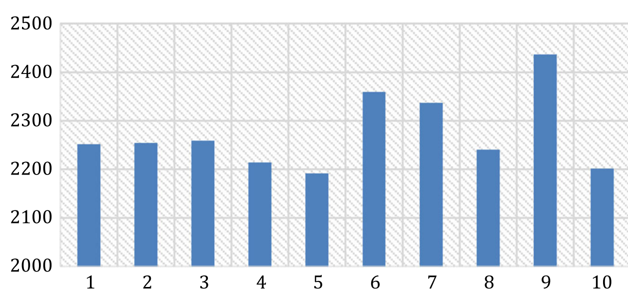

Fig. 4 Variation of static pressure at impeller outlet

represents variations of head at different pressure rise. It reveals that there is marginal difference in the pressure head of both blowers, except one location in an overall range of experimentation.

The graph of efficiency against pressure rise is plotted as shown in Fig. 8. It reveals that efficiency of present design is comparable high in the initial region of pressure rise whereas; at higher pressure rise there is a marginal difference observed in both designs. At duty load conditions, the present design blower gives better efficiency nearly 8% higher compared to the industrial blower. As, efficiency is the ratio of power developed by pressure head to actual

Fig. 5 Pictorial view of manufactured blowers

Table 3 Dimensional details of designs

Dimension	Supplier design blower 1	Present design blower 2
d_2, m	0.45	0.46
d_1, m	0.2	0.2
β_1, degree	35°	24°
β_2, degree	35°	22.8°
b_1, m	0.07	0.07
b_2, m	0.04	0.07
Z	10	8
b_{volute}, m	0.125	0.14

Fig. 6 Experimental rig of CF blower as per IS: 4894-1987

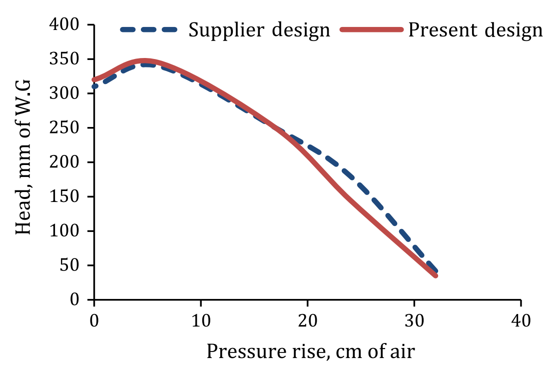

Fig. 7 Head with pressure rise

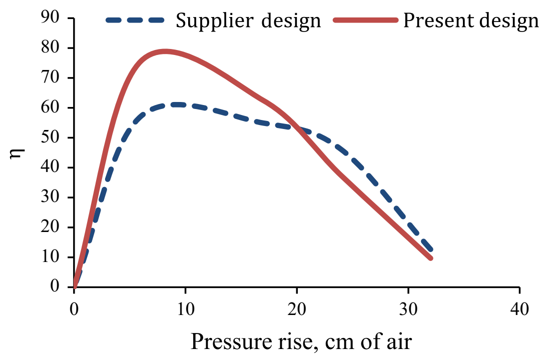

Fig. 8 Efficiency against pressure rise

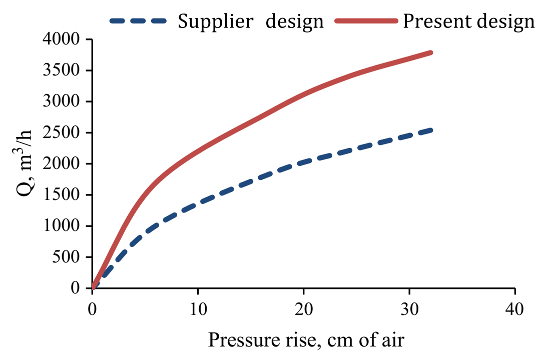

Fig. 9 Flow rate against pressure rise

power consumption, a better efficiency suggests less energy consumption in case of present design.

From Fig. 9, it is observed that the flow rate of present design is higher than the supplier's design throughout the experimental range. This is due to constant width impeller of present design whereas; the supplier's design has tapered impeller. As pressure rise increases the difference in flow rate increase. At duty point, 35% higher flow rate observed with the present design compared to industrial one. The results suggest that the flow rate in straight impeller is higher as it accommodates higher flow compared with tapered impeller for the same pressure rise and gives better efficiency.

Conclusions

The present design methodology based upon unification of principles of various design methodologies subjected to analytical analysis. Following conclusions were drawn from the present work:

(i) Numerical simulation results suggest that the present design is a feasible design with minimum losses, as less pressure variations near the impeller periphery.

(ii) Experimental results indicates better performance in terms of higher flow rate and high efficiency for the same pressure rise in the present design blower compared to industrial one, especially at duty point.

References

1. CERF/IIEC-Asia, *Study of Pumps and Fans Market in China, Lawrence Berkeley National and American Council for Energy Efficient Economy* (CERF/IIEC-Asia, Bangkok, 2002)
2. A.H. Church, Centrifugal Pumps and Blowers (Wiley, New York, 1962)
3. B. Eck, *Fans Design and Operation of Centrifugal, Axial and Cross Flow Fans* (Pergamon Press, Oxford, 1972)
4. W. Osborne, *Fans* (Pergamon Press, New York, 1982)
5. G Yan, D Pan, L Ziang, Parameterized design of centrifugal fan's impeller, International Conference on Measuring Technology and Mechatronics Automation, (2010)
6. G Yan, P Xi, Study on methods of determining main geometric parameters of fan impeller, vol. 4, IEEE, (2010)
7. B Mishra, Design of radial tipped centrifugal blower, M.E. Dissertation Report, (V.N.S.G.U., Surat, 1997)
8. T. Meakhail, S.O. Park, *A Study on Impeller Volute Diffuser Interaction in a Centrifugal Fan*, vol. 127 (ASME, New York, 2005)
9. Masutage, Three dimensional numerical analysis of backward, forward curved radial tipped blade centrifugal fan with varying number of blades, M.Tech. Thesis, (2011)
10. F. Bleier, *Fan Handbook* (McGraw Hill, New York, 1997)
11. B. Baloni, S. Channiwala, V. Mayavanshi, Pressure recovery and loss coefficient variations in the two centrifugal blower volute designs. Appl. Energy **90**(1), 335 (2012)
12. B D Baloni, Studies on volute design philosophies of a centrifugal blower, Ph.D. Thesis, (S.V.N.I.T., Surat 2013)
13. E. Ayder, R. Van den Braembussche, J.J. Brasz, Experimental and theoretical analysis of the flow in a centrifugal compressor volute. ASEM J. Turbomach. **115**(6), 582–589 (1993)
14. S Yang, F Kong, B Chen, Research on pump volute design method using CFD, Int. J. Rotat. Mach. Art ID 137860, **7** (2011). doi:10.1155/2011/137860
15. I Mishina, I Gyobu, Performance investigations of large capacity centrifugal compressor, ASME paper no. 78-GT-3, (1978)
16. E.C. Bacharoudis, Parametric study of centrifugal pump impeller by varying the outlet blade angle. Open Mech. Eng. J. **2**, 75 (2008)
17. D. Adler, Y. Levy, A laser doppler investigation of the flow inside a backswept, closed, centrifugal impeller. J. Mech. Eng. Sci. **21**(1), 1–6 (1979)
18. C.K. Huang, M.E. Hsieh, Performance analysis and optimized design of backward curved airfoil centrifugal blowers. HVAC & R Res. **3**(3), 461 (2009)
19. S.M. Yahya, *Turbines, Compressors and Fan* (Tata McGraw Hill, New Delhi, 2002)
20. S. Wang, L. Zhang, Z. Wu, H. Qian, Optimization of Centrifugal Fan with Different Blade Number and Outlet Angle (Department of Power Engineering, North China Electric Power University, Baoding, 2009)
21. O.P. Singh, T. Sreenivasulu, M. Kannan, Parametric study of centrifugal fan. Int. J. Adv. Eng. Technol. **1**, 33–50 (2011)
22. X. Qiu, D. Japikse, J. Zhao, M.R. Anderson, Analysis and validation of a unified slip factor model for impellers at design and off-design conditions. J. Turbomach. **133**(4), 041018 (2011)

23. A. Whitfield, N.C. Banes, *Design of radial turbomachines* (Longman Scientific & Technical Publications, England, 1990)
24. D M Sharma, N N Vibhakar, Experimental and analytical investigations of slip factor in radial tipped centrifugal fan, International Conference on Mechanical Engineering 2003
25. T. Wright, *Fluid Machinery Performance, Analysis and Design* (CRC Press, Boca Raton, 1999)
26. N N Vibhakar, Studies on radial tipped centrifugal fan, Ph.D. Thesis, V.N.S.G.U. Surat, (2013)
27. L. Chunxia, W.S. Linga, J.Y. Akuib, The performance of a centrifugal fan with enlarged impeller. Energy Converg. Manag. **52**(8–9), 2902–2910 (2011). **Elsevier**
28. B. Baloni, Y. Pathak, S. Channiwala, Centrifugal blower volute optimization based on Taguchi method. Comput. Fluids **112**, 72–78 (2015)