Review Article On Performance Improvement Of IND 25 Centrifugal Fan By Changing Type Of Blade

Vijay Rajpura
Head of R&D Department,
Indabrator A division of Nesco Ltd.,
Karamsad, Gujarat-38832, India

Vivek Brahmbhatt
P.G. Student,
Mechanical Department,
Sardar Vallabhbhai Patel Institute of Technology,
Vasad, Gujarat-388306, India

Gaurav Patel
Assistant Professor,
Mechanical Department,
A. D. Patel Institute of Technology,
New V .V. Nagar,
Gujarat-388121, India

Abstract

The centrifugal fan simulated using computational fluid dynamic (CFD) approach. Flow in centrifugal impeller is simulated by navier-stockes eq. And performances curved are obtained. As a first step, an experimental setup was developed and prototypes of fans were made to carry out measurements of flow and next, a computational fluid dynamics model was developed for the above setup and the results are validated with the experimental measurement. Fine mesh is generated for impeller blade zone to capture the complex flow behaviour inside blade and mesh independency test is carried out for whole computational domain. The MRF (moving reference frame) applied in the CFD analysis of centrifugal fan as a rotating region around the impeller and component of the impeller stationary. Now, performance curved is obtained under the different variable inlet parameter like volume flow rate, rotational speed, flow coefficient and efficiency. The parameter considered is number of blade, inlet and outlet angle, diameter ratio. It observed that number of blade increase, so that reduce in flow passage and more enlarged flow developed. Finally, the result of this numerical analysis to improve the performance of the centrifugal fan.

Keyword: Centrifugal fan, Radial Impeller, Backward Impeller, CFD (Computational Fluid Dynamics), Fan Performance.

1. Introduction

Fans are classified in two categories depend on the direction of air flow: Radial Flow and Axial flow Centrifugal Fans.

The performance efficiency of all centrifugal fans that utilize radial-flow impellers depends greatly on the mode of impelling. This work therefore focuses on different designs of two of the most important parts of the fan– the impeller and the volute casing – together with the evaluation of their operational performances.

In all the designs, fluid enters the inlet port at the center of the rotating impeller which is the suction eye. As the impeller spins, it thrusts the fluid outward radially, causing centrifugal acceleration. As it does this, it creates a vacuum in its wake, drawing even more fluid into the inlet. The volute casing performs the function of slowing the fluid and increasing the pressure. The impeller, driven by the fan shaft adds the velocity component to the fluid by centrifugally casting the fluid away from the impeller vane tips.

The amount of energy given to the fluid is proportional to the velocity at the edge or vane tip of the impeller. Whereas two designs of the volute casings done were essentially the same, differing only on the direction of impelling; seven different impellers were designed and developed. The differences in the impeller designs are not only attributable to the vane profile but also on whether the impeller is open or close. The performance evaluation carried out with the aid of an anemometer revealed that the closed impeller with backward curved vanes has the best performance cum efficiency with respect to the output speed and flow rate. The designed and developed closed-impellers include backward and forward curved vanes while the open-impellers comprise of backward and forward curved; backward and forward inclined; and radial vanes.

The best method of improving the performance of the fan is to understand exactly the three dimensional turbulent flow in the centrifugal fan. The performance curves of a turbo machine can be obtained by theory, computation, and by series of experiments. The experimental analysis is difficult, costly and time consuming. To evaluate predicted performance of theoretical design, various computational methods are available. They offer optimum design solutions without actual fabrication or making prototypes which save time and expenditure. The CFD part is used for improvement the results of Static Pressure generated at the entry to the impeller, static efficiency. The CFD optimization also helped to improve the flow pattern through the centrifugal fan system.

The numerical procedure thus developed requires three functional input parameters volume flow rate (Q, cfm), static pressure, and fan speed (rpm) which gives the output parameters such as number of blades, static efficiency, total efficiency, velocities at entry and exit of the impeller and the entry and exit angles of the blades. Also, the numerical design gave the details of volute casing. The most important and initial step in numerical simulation is geometry definition and grid generation of computational domain. This process includes selection of grid types, grid refinements and defining correct boundary conditions.

Atre Pranav C. et al [1] the numerical procedure is been developed for the high efficiency fan impellers having airfoil blades and the volute casing for it. The airfoil blades are considered to have backward inclined orientation.

The CAD modelling is divided into three parts viz.: (1) Modelling of airfoil blade, (2) Modelling of fan impeller and (3) Modelling of volute casing. The CAD model for whole centrifugal fan system is prepared in CAD package as shown in figure-3

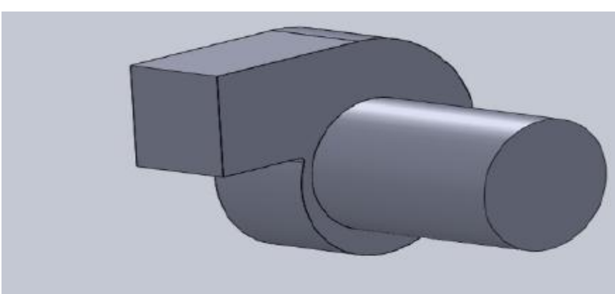


Figure-3 CAD for Centrifugal Fan system

The detailed CAD model is prepared in CAD packages and is meshed using two different software for surface as well as for volume meshing respectively. The mesh details are as given in Table-4.

Table-4 Final mesh details

Entity	Value
Total number of elements	3216076
Maximum cell skewness	0.86

The final meshed model is imported in the CFD solver fluent for pre-processing in which the case setup is carried out. The details of the case setup are given below. Fluent software is used for solving the Navier-Stokes equations governing the physics of the flow inside the centrifugal fan system. Fluent code is based on finite volume method. The Fluent is been used for pre-processing, solving and post-processing purpose.

The three parameters static pressure (SP in inches of water column), static efficiency, power consumed by fan obtained from the numerical design and CFD analysis are correlated successfully for the case undertaken. Hence it can be concluded that the CFD optimization of volute casing helps numerical procedure in improvement of results. However, the variation of 8-9% is observed due to the assumptions in preparing the numerical procedure and CAD model for it. The following conclusions are obtained from the study.

R Ragoth Singh et al [2] in this paper, Taguchi orthogonal array (OA) based design of experiments (DoE) technique determines the required experimental trials. The experimental results are justified by Analysis of Variance (ANOVA) and confirmed by conformation experiments. The parameters chosen for design optimization are Impeller outlet diameter, Impeller wheel width, Thickness of blade and Impeller inlet diameter. The levels for the parametric specification are chosen from the ranges where the blower will get the best efficiency. CFD results were validated by the fine conformity between the CFD results and the experimental results. Optimization of design parameters using this technique is directly inclined towards economic solution for the turbo machinery industry. It has been shown that impeller dimensions were significantly improving the performance of blower by conducting experiments at the optimal parameter combination and also by analyzing S/N ratio.
The contributions of all the design parameters have good importance for determine the performance. The conformation experiments were also conducted to verify the optimal combination of design parameters obtained. Good agreement between the predicted and actual values for static pressure and discharge has been observed.

A. T. Oyelami et al [3] in this paper the performance efficiency of all centrifugal blowers that utilize radial-flow impellers depends greatly on the mode of impelling. Two designs of the volute casings done were essentially the same, differing only on the direction of impelling; seven different impellers were designed and developed. The differences in the impeller designs are not only attributable to the vane profile but also on whether the impeller is open or close. The performance evaluation carried out with the aid of an anemometer revealed that the closed impeller with backward curved vanes has the best performance cum efficiency with respect to the output speed and flow rate. The designed and developed closed-impellers include backward and forward curved vanes while the open-impellers comprise of backward and forward curved; backward and forward inclined; and radial vanes. The impeller speed, which determines the suction rate, was measured at different points, for different vane types/ profiles, from the outlet/discharge of the blower and the results are as shown in Table 1.

Table 1 Blower Performance Characteristics

S/N	Vane Type	Vane Profile	Impeller Type	Speed (m/s) at various distances (0m – 5m) from the outlet										
				0m	0.5m	1m	1.5m	2.0m	2.5m	3.0m	3.5m	4.0m	4.5m	5.0m
1.	Backward Curved		Open	58.9	43.8	34.7	26.5	20.1	18.8	16.3	15.2	13.4	11.8	9.9
2.	Forward Curved		Open	49.6	41.8	29.0	21.7	17.6	14.8	12.2	10.7	9.2	8.1	7.2
3.	Backward Curved		Closed	63.8	49.9	38.3	30.2	23.2	21.5	19.1	17.4	15.1	14.5	12.7
4.	Forward Curved		Closed	54.7	45.2	34.8	24.6	19.8	17.9	15.2	13.2	11.1	9.7	8.6
5.	Backward Inclined		Open	56.3	39.1	29.3	23.3	18.2	17.7	15.8	14.8	12.5	11.2	9.5
6.	Forward Inclined		Open	50.7	40.3	27.9	19.6	17.1	15.0	12.1	9.9	8.7	7.8	7.0
7.	Radial		Open	53.2	41.7	28.1	20.9	17.0	16.4	12.4	11.8	10.7	9.4	9.1

The performance differences in the seven (7) impeller designs are not only attributed to the vane profile but also on whether the impeller is open or close. The closed impeller with backward curved vanes has the best performance cum efficiency with respect to the output speed and flow rate.

N. Yagnesh Sharma et al [4] in this paper performance of the centrifugal fan could be enhanced by judiciously introducing splitter vanes so as to improve the diffusion process. An extensive numerical whole field analysis on the effect of splitter vanes placed in discrete regions of suspected separation points is possible using CFD.

This paper examines the effect of splitter vanes corresponding to various geometrical locations on the impeller and diffuser. The analysis shows that the splitter vanes located near the diffuser exit improves the static pressure recovery across the diffusing domain to a larger extent. Also it is found that splitter vanes located at the impeller trailing edge and diffuser leading edge at the mid-span of the circumferential distance between the blades show a marginal improvement in the static pressure recovery across the fan. However, splitters provided near to the suction side of the impeller trailing edge (25% of the circumferential gap between the impeller blades towards the suction side), adversely affect the static pressure recovery of the fan. Splitter vanes provided on impeller and diffuser at judiciously chosen locations tend to improve the performance of the centrifugal fan, in terms of higher static pressure recovery coefficients and reduced total pressure loss coefficients.

A splitter vane provided at the diffuser trailing edge at the circumferential mid-span between two diffuser vanes (configuration S4) provides relatively large static pressure recovery of the fan. A splitter vane provided near to the suction side of the impeller at a circumferential distance of 25% between the impeller trailing edge (configuration S2) adversely affects the fan performance in terms of static pressure recovery.

Xiaomin Liu et al. [5] in this paper centrifugal impeller firstly by solving the Navier-strokes with the spalart allmaras turbulence model and the performance curves are then obtained. And effect between the inlet duct and impeller inlet on the performance of the centrifugal fan is studies numerically. According to the calculated result, the linkage profile between the inlet duct and impeller inlet is redesigned to improve the performance of the fan. It seen that total pressure and the efficiency increase compared with original centrifugal fan. Then the flow in an inlet duct, an impeller, a diffuser and a volute is analyzed compared with an impeller, the efficiency of the centrifugal fan drop by about 3-4% because of flow loss in the volute. Effects of a straight shroud with different inclined angle, the performance of the centrifugal fan increased and otherwise decreases.

O. P. Singh et al. [6] discussed about in this paper, effect of geometric parameters of a centrifugal fan with backward- and forward-curved blades has been investigated. The parameters considered in this study are number of blades, outlet angle and diameter ratio. And also Effect on the vehicles mileage due to the use of forward and backward fan is also discussed. Figure 5 show that (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 from 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. The results show that 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.

The results suggest that fan with different blades would show same performance under high pressure coefficient. Increase in the number of blades increases the flow coefficient and efficiency due to better flow guidance and reduced losses. The efficiency of the fan first increases and then decreases with diameter ratio. The best efficiency of the fan was observed to be at dia. ratio of 0.5.

S. Ziaei Rad et al [7] in this paper publish paper on “Finite element, Modal testing and Modal analysis of a radial flow impeller” This paper is concerned with the finite element analysis and model testing of an industrial radial flow impeller. The goal is to determine and verify the vibration characteristic of the impeller using both experimental and analytical techniques. First of all FE model of the impeller with tapered blades was built using a 3D element as shown in below fig.

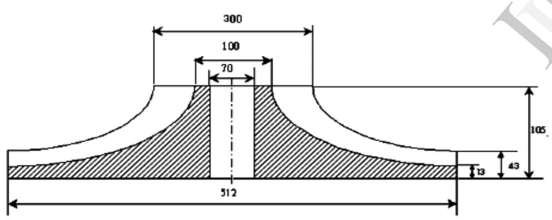


Figure 2.5 Radial flow impeller (in mm)

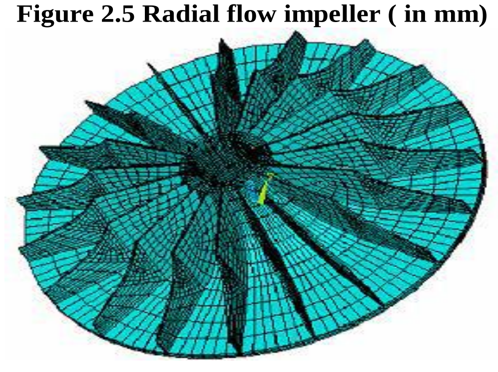


Figure 2.6 Impeller mesh using solid element

The constructed FE model consists of 14256 elements and 114048 DOFS as shown in figure-2.6

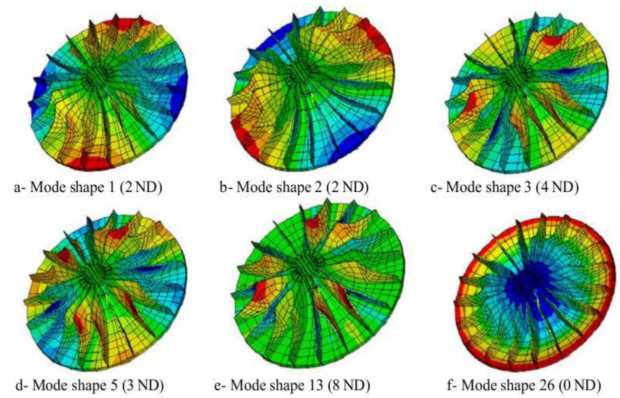


Figure 2.7 the mode shapes of impeller using solid elements

Figure-2.8 plots graph of impeller natural frequencies versus nodal diameters than Campbell diagram for the model in the operating range 0-1800 rev/min was calculated.

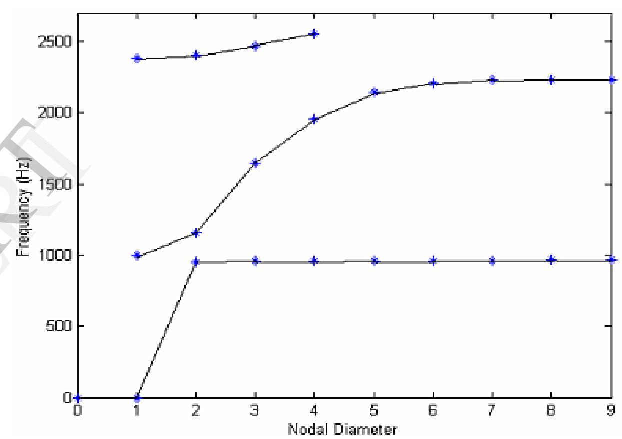


Figure 2.8 Natural frequencies versus nodal diameter

Then the requirement on the convergence of the model must be satisfied. This means that the dynamic properties predicted by the model can be trusted in the frequency range of interested. Two techniques were used to determine the convergence range or the number of converged – predicted modes of the FE models.

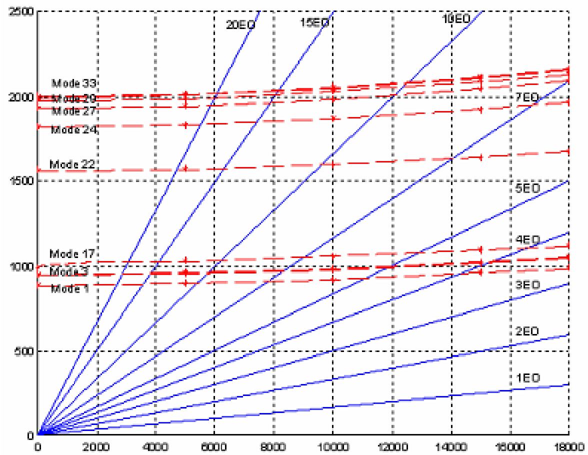


Figure 2.9 the Campbell diagram for the impeller

In the first technique is based on mesh refinement. Two different meshes are applied to the same structure where two model data sets can be obtained. As shown in below figure, are the natural frequency different meshes which were applied in the FE model of the impeller. The model has been converged in the frequency range of interest.

Second technique, two different mass distribution methods were applied for the normal mode solution of each FE model so that two model data sets are obtained for each FE model.

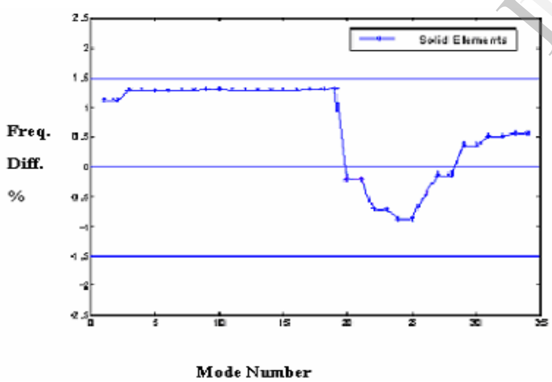


Figure 2.10 Natural frequency differences with different mesh densities

The natural frequency differences between the same modes when the two mass distribution methods, namely consistent and lumped mass distributions were applied. The maximum differences in natural were applied. The maximum difference in natural frequency in the frequency range of interest (0-2300Hz) for the model using solid elements is less than 6.2%.

In pretest planning, first of all the optimum suspension point selection can be done. The suspension arrangements should be considered to make sure that the test structure is supported in the desired condition. Soft springs were used to connect the test structure to ground [8, 9, 10, 12, and 13].It can be assumed that there is no mass, but only stiffness attached to the suspension point. So that any additional forces will result from displacement of the suspension point then optimum driven point is selected.

Next step is the modal testing and modal analysis. Hammer testing: the impeller was suspended by three soft springs and tested in a free-free condition. The experimental data were obtained via hammer testing using a B and k 2032 analyzer connected to a computer. One B&K piezoelectric accelerometer was attached to point 1 on the structure by using wax as shown in below figure.

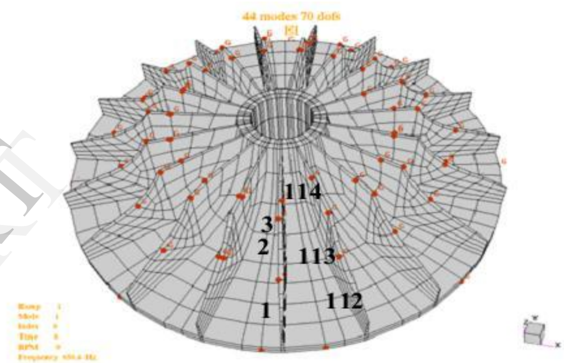


Figure 2.11 a total 114 response points selected.

In model analysis procedure, the experimental FRF data were analyzed using a global multi FRF analyzed using a global multi FRF analysis method. The technique is based on a complex singular value decomposition of a system matrix expressed in terms of measured.

Following table represents the measured natural frequencies for the impeller versus calculated result from FE models.

Measurement			Finite element		Difference
Natural frequency (Hz)	Nodal diameters	Damping %	Natural frequency (Hz)	Nodal diameters	Natural frequency (%)
957.0	2	0.31	950.4	2	0.7
965.5	9	0.99	963.6	9	0.2
969.9	9	0.34	963.6	9	0.7
1004.8	1	0.40	999.6	1	0.5
1024.2	1	0.21	999.6	1	2.4
1151.9	2	0.10	1156.2	2	-0.4
1197.3	2	0.07	1156.2	2	3.4
1311.5	2	0.36	Not Predicted	-	-
1312.6	2	0.30	Not Predicted	-	-
1859.8	3	0.34	1649.7	3	11.3
1862.9	3	0.43	1649.7	3	11.4
2156.9	4	0.37	1952.7	4	9.5
2159.0	4	0.29	1952.7	4	9.6
2323.4	5	0.27	2145.3	5	7.7
2324.5	5	0.41	2145.3	5	7.7
2410.9	5	0.39	Not Predicted	-	-
2610.7	6	0.42	2207.5	6	15.4
2612.2	7	0.13	2228.8	7	14.7
2613.2	7	0.23	2228.8	7	14.7

Table 2.3 Natural frequency results for the impeller (Measured against FE)

Laser Doppler Vibrometer (LDV) measurement, a complete characterization of the mode shapes of an impeller may require a total displacement, velocity/ acceleration field measurement such as holography. But for many bladed-disk applications, a circular line scan near the periphery can be very effective, enabling the back plate mode to be described in terms of its nodal diameter component.

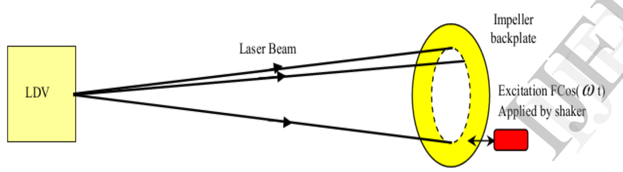


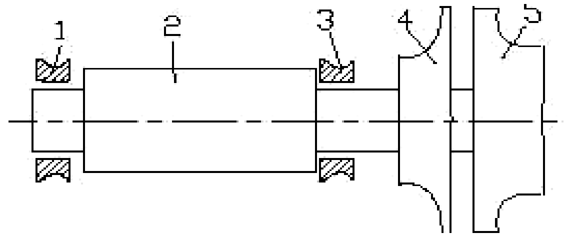
Figure 2.12 Scanning laser Doppler vibrometer applied to impeller back plate

Effect of varying disc thickness

The disc thickness was changed from 0.2 to 4 times the normal thickness. It was found that the variation is not significant for the lower modes. However, significant shifts in the natural frequencies were observed in higher modes. The maximum variation in natural frequency is around 38 percent for modes in the frequency range of 1400 to 1600Hz.

Liangwei Zhong et al [8] in this paper the technique of three dimensional solid element model and assembly was used to determine the temperature field of radial-flow impeller.

The FEM software Cosmos was applied to analyze model system, and the precise analytic results were obtained.



1.3 bearing 2.motor rotor 4. Compressor blade 5. Turbo-expander wheel

Diagram 1.Structural diagram of gas turbine

The analytic model and the computing method of the steady and transient temperature field of blade wheel rotor the first law of thermodynamics that is the law of conservation of energy, we can have the following equation:

$$Q - W = \Delta U + \Delta KE + \Delta PE \quad (1)$$

In this equation,
Q ——heat
W —— work
ΔU ——internal energy of a system
ΔKE ——kinetic energy of a system
ΔPE ——potential energy of a system

As long as most of the engineering thermal transmission problems are concerned:

$$\Delta KE = \Delta PE = 0$$

In Steady thermal analysis, steady to transient is dealt with, the steady thermal analysis should be taken as the last step of the transient thermal analysis and the condition of the system in the steady can be determined. The equation of energy balance of the steady thermal analysis can be expressed in the form of matrix like this:

$$[K]\ \{T\} = \{Q\}$$

In Transient thermal analysis, the temperature field is changing with time. As long as the problem of un-periodic thermal transmission is concerned, the temperature inside the objects going up or down constantly with time will go to the temperature of the environment and achieve the balance ultimately after a long time.

K. Vasudeva Karanth et al. [9] Study about the Effect of Radial Gap on impeller-diffuser flow of a centrifugal Fan figure 3. The flow between the impeller exit and the diffuser entry. With the development of PIV and CFD tools such as moving mesh techniques and numerical methodology involving moving mesh technique is used in predicting the real flow behaviour, as exhibited when a target blade of the impeller is made to move past corresponding vane on the diffuser.
Result found that there is an optimum radial gap at which better dynamic and static heads are developed by the impeller blades as well as better

energy conversion by diffuser vanes and maximum efficiency of the centrifugal fan as observed in the study.

My Coments

From the above review it is conclude that the following scope of work.

- Volute Casing Optimization
- Change blade shape for performance improvement
- Blade Geometry Optimization
- CFD analysis for Higher order Model
- Fluid Structure Interaction analysis

Conclusions

In this paper, an investigation on the effect of centrifugal fan parameters on performance has been presented thorough experiments and a CFD simulation has been presented. Test results show that Increase in the number of blades increases the flow coefficient accompanied by increase in power coefficient. Increase in the number of blades increases the flow coefficient and efficiency due to better flow guidance and reduced losses. The best efficiency of the fan was observed.

References

[1] Atre Pranav C., Thundil Karuppa Raj R., "Numerical design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impeller", *Research Journal of Recent Sciences, Vol.1 (10) 7-11, October 2012*.

[2] R Ragoth Singh, M Nataraj, "Optimizing Impeller Geometry for Performance Enhancement of a Centrifugal Blower Using The Taguchi Quality Concept", *International Journal of Engineering Science and Technology, Vol. 4 No. 10, October 2012.*

[3] A. T. Oyelami, S. B. Adejuyigbe, A. K. Ogunkoya, "Analysis of Radial-Flow Impellers of Different Configurations", *The pacific Journal of Science and Technology, Vol. 13 No.1, May 2012.*

[4] N. Yagnesh Sharma, K. Vasudeva Karanth, "Numerical Analysis of a Centrifugal Fan for Improved Performance using Splitter Vanes", *World Academy of Science, Engineering and Technology Vol.36, 2009.*

[5] Xiaomin Liu, Qun Dang, Guang Xi, "Performance Improvement of Centrifugal Fan by Using CFD", *Engineering Applications of Computational Fluid Mechanics, Vol. 2 No.2, pp.130-140, 2008.*

[6] O. P. Singh, Rakesh Khilwani, T. Sreenivasulu, M. Kannan, "Parametric Study of Centrifugal Fan Performance: Experiment and Numerical Simulation", *International Journal of Advances in Engineering & Technology, Vol. 1, Issue 2, pp.33-50, May 2011.*

[7] S. Ziaei Rad, Finite Element, "Model Testing and Model Analysis of a Radial Flow Impeller", *Iranian Journal of Science & Technology, Transaction B, Engineering, Vol. 29, No. B2, 2005.*

[8] Liangwei Zhong, Kangmin Chen, Jing Ni, Anyang He, "Three Dimensional Finite Element Analysis of Radial Flow Impeller Temperature Field", *Modern Applied Science Vol. 2, No. 6, November 2008*.

[9] K. Vasudeva Karanth and N. Yagnesh Sharama, "CFD Analysis on the Effect of Radial Gap on Impeller-Diffuser Flow Interaction as well as on the Flow Characteristics of a Centrifugal Fan", *International Journal of Rotating Machinery, Volume 2009, Article ID 293508, 8 Pages.*

[10] S. M. Yahya, Turbines, Compressors and Fans, Tata McGraw-Hill Publishing Company Limited, 2005.

[11] Nicholas P. Cheremisinoff, Pump, Compressors and Fans, Technomic Publishing Co. INC. Lancaster, 2000.

[12] ECK Bruno, Fan- Reference book on fan engineering, 2nd edition, Pergamon press, 1977.