Research Paper Engineering

# A Review on Performance of Industrial Induced Draft Centrifugal Fan Using Ansys Cfx

* J.I. Patel ** R.N. Makadia

* P.G. Student, School of Engineering, RK University, Rajkot, Gujarat, India

** Assistant Professor, School of Engineering, RK University, Rajkot, Gujarat, India

**ABSTRACT**

*Fans are one of the types of turbo machinery which are used to move air continuously with slight increase in static pressure. Fans are widely used in industrial and commercial applications from shop ventilation to material handling, boiler applications, transporting gas or materials and most use in the HVAC industry today. In this paper it describes the different techniques which may help to increase the pressure of air. Flow in centrifugal impeller is simulated by Navier-stokes eq. and performance curved is obtained. The experimental data has been collected from Industry. Now the parameter considered is number of blade, inlet and outlet angle, diameter ratio Performance analysis has been carried out by experimental and ANSYS CFX software. It observed that number of blade increase, so that reduce in flow passage and more enlarged flow developed. Finally, the result of this total pressure and efficiency increase compared with original centrifugal fan.*

**Keywords : Centrifugal fan, Types of Blade, low cost, sustainable**

**INTRODUCTION**

Centrifugal fans widely used in various fields of engineering. With the expansion of its use in industry, the centrifugal fan is developed towards the direction of high efficiency.

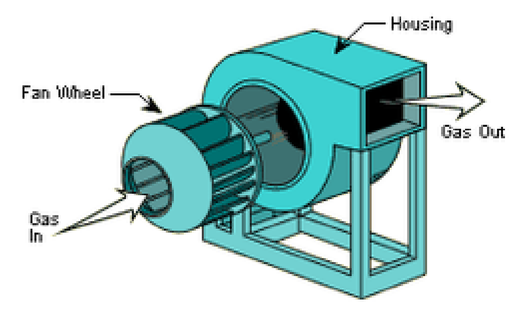


**Figure 1: Schematic diagram of Centrifugal fan**

Main parts of a centrifugal fan are Fan Housing, Impellers, Inlet and outlet ducts, Drive Shaft, Drive mechanism. It has a fan wheel composed of a number of fan blades, or ribs, mounted around a hub. As shown in Figure 1, the hub turns on a drive-shaft that passes through the fan housing. The gas enters from the side of the fan wheel, turns 90 degrees and accelerates due to centrifugal force as it flows over the fan blades and exits the fan housing. They are study, quiet, reliable, and capable of operating over a wide range of Conditions. Centrifugal fans are capable of generating relatively high pressures. They are frequently used in "dirty" air steams (high moisture and particulate content), in material handling applications and in system operated at higher temperatures. In industrial application fans are commonly used to supply ventilation or combustion air, to circulated air or other gases through equipment and the exhaust air or other vapours from equipments.

**Types of Blade:**

There are many types of Fans.

1. Forward-Curve blades, $\beta_2 <$
2. Backward-Curve blades, $\beta_2 <$
3. Radial blades, $\beta_2 = 0$

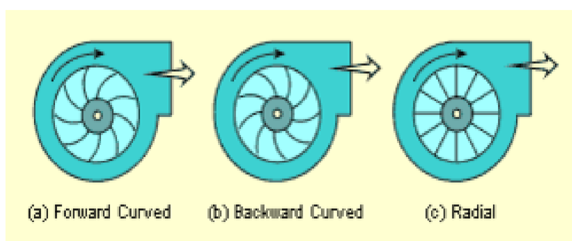


**Figure 1: Blade curve.**

**Atre Pranav c. et al .[1]**The following paper presents the design methodology for the centrifugal fan system with impellers having airfoil blades. The numerical design procedure is developed for it and the CFD optimization has been carried out for volute casing to improve the results which have got from the numerical procedure only. The volute casing was optimized by decreasing the volute clearances by 10-14% and increasing the cut off height by 5% keeping it at 35% of impeller diameter. The cut off height plays the major role in prevention of recirculation occurring near outlet region of the fan system. the design methodology thus developed for high efficiency centrifugal fan impellers with airfoil blades which includes numerical design as well as the CFD parametric optimization of Volute casing has been successfully implemented and validated.

**Li Chunxi et al .[2]**G4-73 type centrifugal fan performance is investigation. comparison are conducted between the fan with original impeller and two larger impellers with the increments in impeller outlet diameter of 5%and 10%respectively in the numerical and experimental investigations. Comparisons between experiment results and the trimming laws show that the trimming laws for usual situation can predict the performance of the enlarged fan impeller with less error for higher flow rate, although the situation of application is not in agreement. The present paper investigates the effect of blades tip extension and impeller enlargement on centrifugal fan performance. Numerical simulations and experiments of the original impeller and enlarged impellers are conducted and the results are compared.

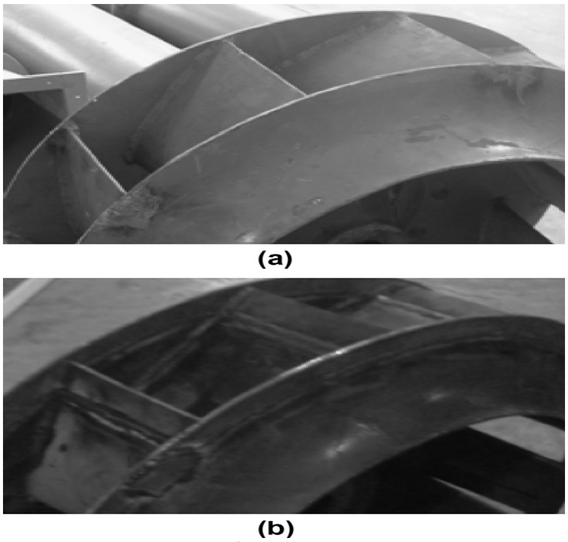


**Figure 3: The photos of original impeller and the 880 mm diameter impeller.(a) The original impeller.(b) The enlarged impeller with blades tip extended.**

**Abolfazl khalkhali et al .[3]** In the present study, multi-objective optimization of forward curved blade centrifugal fan is performed. Head rise and head los in a set of forward curved centrifugal fan is numerically investigated using commercial software NUMACA .Genetic algorithms have been used both for optimal design of generalized GMDH type neural network models of Head Rise and Head Loss in forward curved centrifugal fans and for multi-objective pareto based optimisation.

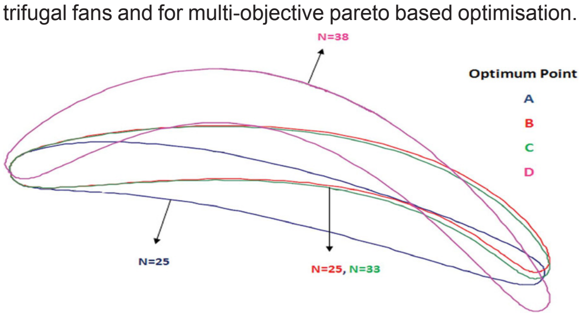


**Figure 4: Overlay graph of the design variables in optimum points.**

**R RAGOTH SINGH et al .[4]** presented the methodology to find near optimum combination of blower operating variables for performance enhancements were analyzed using computational fluid dynamics (CFD). The Taguchi method for optimizing the design parameters in blower operation. Optimization of design parameters using this technique is directly inclined towards economic solution for the turbo machinery industry. Best efficiency find in this paper.

**Tahsin Engin et al [5]**. In the present experimental study, three semi-open centrifugal fan impellers have been designed and fabricated using ceramic materials to provide high resistance to temperature. Experiments have been conducted to investigate the performance characteristics of these impellers and the deteriorations in their performance due to varying tip clearance. Factors have been determined to estimate the tip clearance losses. The effects of impeller geometry shaft speed, gas temperature, and the tip clearance on the overall performance of the tested impellers have been studied experimentally. The typical design is done based on the required pressure rise and flow rate. In order to increase the fan internal efficiency, the following parameters are usually changed without reducing design pressure and flow rate: d1/d2.

**K. Vasudeva Karanth et al. [6]** Study about the Effect of Radial Gap on impeller-diffuser flow of a centrifugal Fan. The flow between the impeller exit and the diffuser entry (i.e., in the radial gap is generally considered to be complex). 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.

**Sheam-Chyun Lin et al.[7]** Conducted preliminary experimental and numerical study of Forward curve centrifugal fan. A small centrifugal fan is successfully designed for the thermal task of cooling laptop computers by utilizing an integrated scheme, which consists of fan design, mock-up manufacture, experimental verification, and numerical simulation. In this research, a cooling fan is designed under the space limitations of notebook computers. Prototypes are manufactured by the computer numerically controlled (CNC) machine to carry out the corresponding experimental verifications. By comparing the experimental and numerical results, a good agreement between them indicates a great potential to reduce expensive experimental work by using CFD tool.

**Sheam-chyun Lin et al. [8]** Study about an integrated performance analysis for a backward-inclined centrifugal fan and performance evaluation of fan design under different operating conditions. Performance analysis for a BI centrifugal fan is carried out through a combined experimental and numerical approach. An 80 mm-diameter backward inclined centrifugal fan is chosen to serve as the research subject for demonstration purposes. Numerical results are utilized to perform detailed flow visualization, torque calculation, efficiency estimation, and noise analysis. The results indicate that the fan performance curve and the sound pressure level (SPL) spectrum of the experiment agree with those of numerical simulations and flow visualization at each operating point, having verified the successful enhancement of fan performance via numerical calculation.

**Xiaomin Liu et al. [9]** 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. [10]** 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. (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. 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 diameter ratio of 0.5.

**N. Vibhakar et al. [11]** experimented on a Backward Curved Radial Tipped Blade Centrifugal Fan. In this study, fan geometry is obtained as per unified design methodology and CFD analysis carried out in this work is to understand the volute-impeller interaction at design and off-design conditions under varying mass flow rates, rotational speeds and number of blades. Steady, realizable k-ε model with MRF approach is used to evaluate the flow behaviour inside centrifugal fan by using ANSYS software. Performance curves are obtained under the Rotational speed of impeller constant and varying, Number of impeller blades as 12, 16 and 24 and Volume flow rate from 0.1 to 0.5 m3/sec. Maximum total efficiency 45.89%, which occurs at 0.25 flow coefficient.

The effect of varying rotational speed while 16 numbers of blades kept constant show that power coefficient and efficiency are almost Unaffected near design low coefficient and has different values at lower and higher flow coefficients. Hence all the quantities varying with flow coefficient are increased as the number of blades increases. The streamlines clearly shows the rotating effect of blades on flow, within and outside of impeller zone, which is imposed using MRF approach. The numerical analysis shows that, for efficient energy transfer i.e. to achieve optimum performance in centrifugal fan designed by unified methodology number of blades should increase.

## 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. and Thundil karuppa Raj R. “Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers”. Research Journal of Recent Sciences. ISSN 2277-2502. vol.1 (10), 7-11, October (2012). | [2] Li Chunxi, wang, jia yakui. “The performance of a centrifugal fan with enlarged impeller”. Energy Conversion and Management 52 (2011) 2902–2910. | [3] Abolfazl Khalkhali, mehdi Frajpoor, Hamed Safikhani. “Modeling and Multi-objective optimization of forward-curved blade centrifugal fans using CFD and neural networks”. Transactions of the Canadian Society for Mechanical Engineering, Vol. 35, No. 1, 2011. Received May 2010, Accepted January 2011.No.10-CSME-27, E.I.C. Accession 3190.| [4] R Rogath Singh. “Optimizing impeller geometry for performance enhancement of a centrifugal blower using the tanguchi quality concept. R Ragoth Singh et al. / International Journal of Engineering Science and Technology (IJEST) ISSN : 0975-5462, Vol. 4 No.10 October 2012. | [5] Tahsin Engin, Mesut Gur , Reinhard Scholz. “Effects of tip clearance and impeller geometry on the performance of semi-open ceramic centrifugal fan impellers at elevated temperatures”. Experimental Thermal and Fluid Science 30 (2006) 565–577. | [6] K. Vasudeva Karanth and N. Yagnesh Sharma. “CFD Analysis on the Effect of Radial Gap on Impeller Diffuser Flow Interaction as well as on the Flow Characteristics of a Centrifugal Fan”. ISSN 1 746-7233, England, UK World Journal of Modelling and Simulation Vol. 5 (2009) No. 1, pp. 63-71. | [7] Sheam-Chyun Lin, Chia-Lieh Huang. “An integrated experimental and numerical study of forward–curved Centrifugal fan”. Experimental Thermal and Fluid Science 26 (2002) 421–434. | [8] Sheam-Chyun Lin, Ming-Lun Tsai. “An integrated performance analysis for a backward-inclined centrifugal fan”. Computers & Fluids 56 (2012) 24–38.| [9] Xiaomin Liu, Qun Dang and Guang Xi. “Performance improvement of centrifugal fan by using CFD”. Engineering Application of Computational Fluid Mechanics Vol. 2, NO. 2. pp 130-140 (2008).| [10] O. P. Singh, Rakesh Khilwani, T. Sreenivasulu, M. Kannan. “Perametric study of centrifugal fan performance: Experiments and numerical calculation”. International Journal of Advances in Engineering & Technology May 2011.ISSN:2231-1963.| [11]. N.Vibhakar, S.D Masutage and S.A.Channiwala. “Three Dimensional CFD Analysis of Backward Curved Radial Tipped Blade Centrifugal Fan Designed as per Unifie Methodology with Varying Number of Blades”. International Journal of emerging trends in engineering and development Issue 2, Vol.1 (Jan-2012) ISSN 2249-6149 Page.