Impact Factor (SJIF): 3.632

International Journal of Advance Research in Engineering, Science & Technology

e-ISSN: 2393-9877, p-ISSN: 2394-2444

Volume 4, Issue 1, January-2017

A Review on Design and CFD Analysis of Casted Centrifugal Fan for Heavy Duty Industrial Applications

Nitesh Patel[1] , Nitesh Panchal[2], Akash Mohanty[1]

[1]*School of Mechanical Engineering, VIT University, Vellore-632014, Tamilnadu, India.*

[2]*Spruce Up Industries Pvt. Ltd., Pune-411048, Maharashtra, India.*

Abstract*– This paper reviews the key plan parts of centrifugal fan and casing for suction based cyclone dust collector for industries, i.e. the determination of a suitable blade geometry profile for the fan blade in order to enhance the energy efficiency of centrifugal fan, utilizing CFD simulation technique. The issue of centrifugal fan choice and configuration usually introduce itself in the accompanying way: a specific air volume and static pressure are required for a certain system. Once the need of fan and casing plan on a theoretical basis has been recognized, it is possible to determinate the measurements for a fan and casing so that it could be fabricated by suitable jigs and fixtures installation and perform as per a specific arrangement details. Design of volute casing, fan and hub is iterative process for better security. Fundamental angle is blade geometry for proper air flow at inlet 6-12 blade generally taken on the grounds that more blades will limit the flow of air. CFD examination determined the air flow from hose inlet to casing outlet, and reviewed the pressure losses predicted by simulation.

Keywords— *centrifugal fan, casing, CFD*

I. INTRODUCTION

Centrifugal fans are the turbo machines broadly utilized as a part of present modern and household life. Recognize that the plan of any turbo machine is an interdisciplinary procedure, which includes aerodynamic features, thermodynamics, fluid dynamic, stress analysis, vibration investigation, the choice of materials, and the prerequisites for assembling. In spite of the fact that centrifugal fans have been exceptionally effective turbo machines, plan is still in view of different experimental and semi observational standards proposed by fan originators. Manufacturing industries of fans from time to time followed optimum design solution for individual fan/blower. For the most part their design and manufacture is based on arrangement of effective past models or got from fan laws and geometrical likenesses. Amid broad writing survey on design and execution assessment of Fan Performance, it is watched that much research work has been done on local flow physics, optimal design and marvels of vitality exchange. Design of these turbo machines requires calculation with numerous factors and coefficients. It is likewise concentrated that the design systems proposed by various researchers vary broadly. It has uncovered lacuna of explicit design approach which can give sought execution. One of the goals of present review is to design and investigate execution of explicit systems as recommended in literature. Three orderly design approaches for centrifugal fan are followed out after extensive literature audit. Concentrating on these three outline techniques, relative evaluation is made numerically and after that analyses are completed to get optimum design solution. These design methodologies are summarized as under and discussed in detail subsequently.

II. RESEARCH AND STUDY ON FAN AND CASING

A. METHODOLOGY TO DESIGN PARAMETERS

This research work is based on an industrial requirement for dust and litter collection fan for street vacuum cleaning machine. Here variable flow is required at constant head under dust laden conditions. Radial blades are ideal for dust laden air or gas because they are less prone to blockage, dust erosion and failure and have self-cleaning properties. Atre and Karuppa-Raj had studied Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers and the design methodology for the fan system with various design of impeller [1]. The results are correlated by a case studied and numerical results obtain. They used the MRF concept (moving reference frame in ansys to simulate rotation of flow around rotating frame impeller by creating flow one. Nataraj and Arunachalam optimizing impeller geometry for performance enhancement of a centrifugal pump using the Taguchi quality concept they determine the results by various

geometry of impeller to enhance the performance of centrifugal blower [2]. To reduce the required no of trail they used OA (orthogonal array technique) and justification is held by them by using (ANOVA) technique. There optimization is focused on following parameter Impeller inlet dia, impeller outlet dia, impeller wheel width, thickness of blade. Mayaing et al. studied and collected input datas from shwe nana ta heat mill factory. Design of shaft, impeller and casing reported in this paper as they are main component of fan system. In this paper, various methods are displayed to simulate the flow. The first step is assigning about computational domain, boundary condition, rotating region. Second step is assigning goal for flow simulation analysis and final step is to run the simulation. Keyur K. Patel, Prajesh M. Patel this paper is about Fine mesh is generated for impeller blade zone to capture the complex flow behavior inside blades 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 [4]. Jayapragasan et al. focused on two types of impeller forward curved and radial type to analyze for standardization with better performance [5]. Examination held for both fan design methodology and performance using CFD analysis by Jayapragasan et al. [6]. They studied the performance parameters such as effect of fan geometry, speed of fan and fillet radius at inlet. By keeping casing and number of blade constant total discharge and fan efficiency is calculated. Procedure followed in this paper to simulate and design of fan casing is 1st Modeled in solid works, 2nd step is the meshing by ICEM Technique and the 3rd step solved in model using ansys fluent v6 and finally to reduce the no of trail and to arrange them properly taguchi method and mini tab id used. Patil et al. explore the work which is given to utilize theoretical investigation in combination with CFD ideas for assurance of optimum estimation of blade angle for most extreme pressure head and release for water pump. Because of radial movement, the air is pushed out of the fan entry and consequently courses through the spiral casing [8]. The spiral casing keeps up the discharge of the flow and guides out the air to outlet. The geometry of the centrifugal fan plays an essential part in their execution. In view of the fan geometry [Fig. 1], it is named as backward curved fan and forward curved fan, and radial fan. The forward curved and in backward curved fans gathers the cushions at focus, and accordingly builds the power.

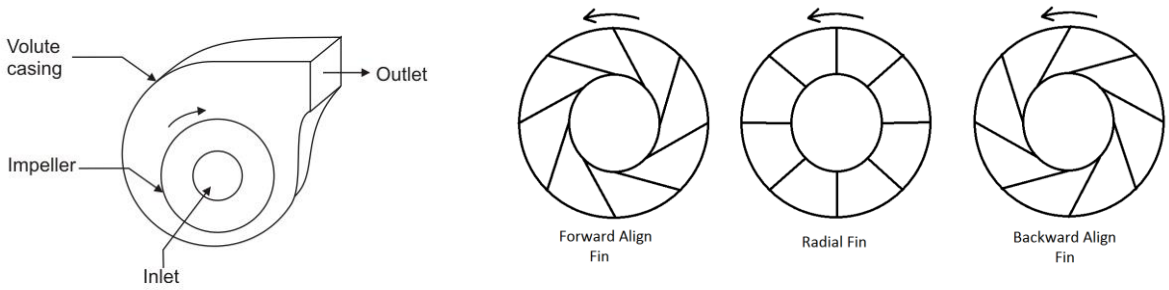

Fig. 1. Impeller and its types [2]

In this review, outspread sharp edge fan was used which is most usually utilized for clean accumulation framework. The parameters for example, volute packaging, inlet fan diameter, tip freedom, and fan leeway are kept consistent during the investigation. The fan external breadth, fan sharp edge point, and number of cutting edges are considered as streamlining parameters which serves to get high static pressure with less power utilization**.** Ramasamy et al., [9] discussed the design impeller is investigated utilizing a CFD approach. The accompanying are the presumptions made while utilizing CFD approach.

1. Incompressible flow,
2. No-slip limit condition,
3. Gravity impacts are irrelevant,
4. Liquid properties are not elements of temperature.

Dhande et al. [10] observed that blade solidity and there spacing effect the performance of blower.in this they study on number of blade in blower fan, by varying no of blade and rpm the enlisted the effect on other parameter as inlet velocity, air discharge, air pressure, power requirement, blower efficiency and performance index.

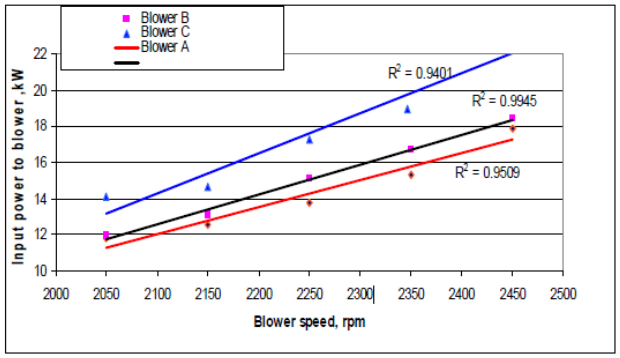

Fig. 2 Input power Vs. Blower speed

The data observed from this research is that when you vary the rpm the total pressure of blower ill increase at different rpm.and percentage of blower also increase as per rpm (Fig. 2). Karanth and Sharma [11] observed that when radial gap is relatively larger, then also the jets and wakes related to the exit flow from the Fan gets evened out, the flow tends to divert from some of the vanes of the diffuser due to change in angle of incidence and this will effects on static pressure increases across the impeller and this will affect the efficiency. This is observed in the way that there is high flow rate compare to design point operation.

B. OPTIMIZATION DESIGN PROCEDURES

Bin et al. [12] described the procedure for blade design and optimization procedure for blade. They followed the step by step performance ability of flow diagram (Fig. 3). 1st requirement is the aerodynamic parameter like total pressure, inlet and outlet flow Angles, flow rate etc. 2nd requirement is the blade geometry and parameterized by breezier curve. The 3rd objective is the functional values and aerodynamic performances of blade sample database which were calculated by CFD solver.

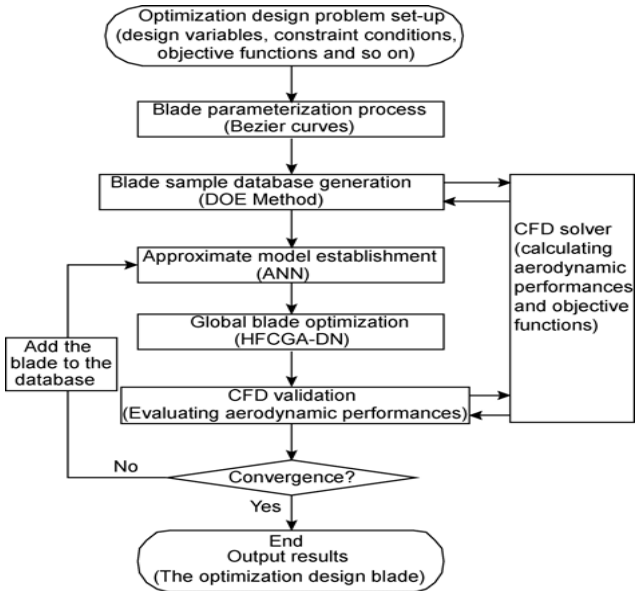

Fig. No. 3 Flow diagram for optimization of blade design.

Three-dimensional Navier-Stokes conditions and vitality condition are explained on body-fitted matrices utilizing an unequivocal limited distinction plot [13]. Thick terms in the stream shrewd heading are disregarded utilizing a thin-layer guess, and those in different bearings are computed. The Baldwin-Lomax model is picked as the turbulence show. An express Runge-Kutta plan is used to walk from the underlying to relentless state with a spatially shifting time venture to quicken meeting. Shojaeefard et al. [14] observed about MRF technique used for flow simulation amd numerical solution validation with experimental results and CFD analysis. Oyelami et al., [15] study concentrates on various outlines of two of the most essential parts of the blower, the impeller and the volute packaging together with the assessment of their operational exhibitions. In every one of the outlines, liquid enters the delta port at the focal point of the pivoting impeller which is the suction eye. As the impeller turns, it pushes the liquid outward radially, creating radiating speeding up. As it does this, it

makes a vacuum afterward, drawing considerably more liquid into the gulf. The volute packaging plays out the capacity of abating the liquid and expanding the pressure. Huang et al. [16] focus on the reenactment plan and demonstrate the substantial; the impacts of sharp edge point, cutting edge number, tongue length, and parchment shape were examined. A few parameter changes are proposed in light of these reenactments. An enhanced configuration is given a 7.9% change in static weight and a 1.5% change in productivity. Cheah et al. [17] aims about simulation of complex internal flow at different flow rate condition. Anagnostopouls [18] observed that number of controllable design variables helps in design shapes of impeller. Croba and Kueny [19] describe about numerical modeling of centrifugal pump and suggested techniques to reduce vibration, radial forces and hydraulic noise. Mentzos et al. [20] predicts that flow pattern, pressure distribution in untwisted blades and he also explains about performance head- curves of impeller blade. Gurupranesh et al. [21] reveals about design modification of impeller which gives moderate head and discharge in required flow condition.

C. OBSERVATIONS

From the above review it was conclude that the

Following points high lights the scope of the present review study:

- CFD analysis of centrifugal fan
- CFD analysis of volute casing
- Optimization of blade angle
- CFD validation

III. CONCLUSION

In this paper, details review about types of centrifugal impellers and there effects on flow parameter static pressure, total pressure, air flow at inlet and outlet was carried out. It was observed that the CFD analysis plays an important role to reveals about distribution of air flow in overall casing and it helps to optimize the blade geometry and casing dimension. The vacuum based cleaning machine is used to extract dust and litter required high vacuum pressure, hereby the fan performance and characteristic curves helps in selection of proper rpm for fan which create required air flow. Finally, the optimization methods applied to fan speed, impeller inlet diameter, number of blades and width of blades etc. Simulation of flow is simulated by using MRF (multi rotating frame) in ansys fluent.

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, pp. 7-11, Oct. 2012.

[2] M. Nataraj and V. P. Arunachalam, "Optimizing impeller geometry for performance enhancement of a centrifugal pump using the Taguchi quality concept", Proceedings of the Institution of Mechanical Engineers, Part A, Journal of Power and Energy 220(7), pp. 765-782 · November 2006.

[3] Kay Thi Myaing, Htay Htay Win, "International Journal of Scientific Engineering and Technology Research, 3(10), pp 2138-2142, May 2014.

[4] Keyur K. Patel, Prajesh M. Patel, "Performance improvement of centrifugal fan by using cfd", International Journal of Advanced Engineering Research and Studies, March 2013.

[5] Jayapragasan.C.N, Manoj kumar.A, Dr. Janardhan Reddy.K, Design Optimization and Parametric Study on the Alternative Blower of Travelling Cleaner, International Journal of Innovative Science, Engineering & Technology, 2015

[6] C. N. Jayapragasan, Sumedh J. Suryawanshi and K. Janardhan Reddy, Design optimization of centrifugal fan for travelling cleaning, Journal of Scientific & Industrial Research October 2016, pp. 638-642

[7] Pranit M. Patil, Sangram S. Patil, Dr. R. G. Todkar, Design, Development and Testing of an Impeller of Open Well Submersible Pump for Performance Improvement. International Journal of Science and Research, 2012

[8]Raj. D, Swim, W.B., Measurement of the Mean Flow Velocity and Velocity Fluctuation at the exit of FC Centrifugal Rotor , Trans ASME, J. ENG. Power: p. 393-399,1981

[9] Haridass RamasamyP P, Prabhu Prakash CFD Approach in the Design of Radial Flow Centrifugal Pump Impeller

[10] K.G Dhande, Ravi mathur and Ajay sharma, Performance Evaluation of a centrifugal blower of air assisted sprayer for orchard pesticide application, International Journal of Agricultural, Science and Research (IJASR), 2016

[11] 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, Hindawi Publishing Corporation International Journal of Rotating Machinery , 8 pages,2009

[12] Zhang Bin, Wang Tong, G U Chuangang and shu xin Wei Blade optimization design and performance investigations of an ultra-low specific speed centrifugal blower, science china technologies, page 203–210, January 2011

[13] O. P. Singh, Rakesh Khilwani, T. Sreenivasulu and M. Kannan. . Parametric study of centrifugal fan performance: experiments and numerical simulation. International Journal of Advances in Engineering & Technology, May 2011.

[14] M.H. Shojaeefard , M. Tahani , M.B. Ehghaghi, M.A. Fallahian , M. Beglari Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid,joural Elsevier, Computers & Fluids 60 page 61–70 , 2012

[15] Adekunle Taofeek Oyelami, Olawale Oluwadare Olaniyan, Dalyop Iliya and Abimbola Samson Idowu the Design of a Closed-Type-Impeller Blower for a 500kg Capacity Rotary Furnace.

[16] Chen-Kang Huang and Mu En hsieh,Performance Analysis and Optimized Design of Backward-Curved Airfoil Centrifugal Blowers, American Society of Heating, Refrigerating and Air-Conditioning Engineers, Inc. (www.ashrae.org). Published in HVAC&R Research, 2009

[17] K.W. Cheah, T. S. Lee, S. H.Winoto, and Z.M. Zhao, Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions. Hindawi Publishing Corporation International Journal of Rotating Machinery, 8 pages, 2007

[18] John's anagnostopouls, CFD Analysis and Design Effects in a Radial Pump Impeller. wseas transactions on fluid mechanics, July 2006

[19] D. Croba and J.L Kueny, Numerical calculation of 2D, unsteady flow in centrifugal pumps: impeller and volute interaction. International journal for numerical methods M mums, 1996

[20] Michalis D. Mentzos 1, Andronicos E. Filios A.E., Dionisios P. Margaris and Dimitrios G. Papanikas, CFD predictions of flow through a centrifugal pump impeller. 1st International Conference on Experiments/Process/System Modelling/Simulation/Optimization 1st IC-EpsMsO Athens, 6-9 July, 2005

[21] P.Gurupranesh , R.C.Radha , N.Karthikeyan, CFD Analysis of centrifugal pump impeller for performance enhancement this paper reveals about design modification of impeller which gives moderate head and discharge in required flow condition. IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) PP 33-41