A REVIEW ON DETERMINATION OF FLOW IN THE INDUSTRIAL BLOWER BY USING COMPUTATIONAL FLUID DYNAMICS

[1]Lalit Tasodia,[2]Dhananjay Yadav,[3]Anil Verma

[1] *M.tech Schlor, Mechanical Department, SSSUTMS, M.P.India*
[2] *Assistant Professor, Mechanical Department, SSSUTMS, M.P.India*
[3] *Assistant Professor, Mechanical Department, SSSUTMS, M.P.India*

ABSTRACT

The blower is among the key components in any industrial applications like boilers, cooling systems, dryers etc. Hence the performance and the characteristics of the blower affect the whole system and the efficiency of the work. The blower system consists of mainly three parts suction hood, impellor and volute which determine the characters of the blower. It is very important to predict the air flow for a blower before its application. A study on the industrial centrifugal blower is done to accurately predict the air flow and mass flow using Computational Fluid Dynamics (CFD) tool. The detailed model of blower is created with the help of a 3D CAD Software and a 3D CFD Fluent code is used to solve different governing equations and predict the flow at outlet of the blower. The numerical results obtained with the help of CFD are compared with the characteristic curve supplied by the manufacturer.

Keywords— *ANSYS, Fluent, CFD analysis;*

INTRODUCTION

Blowers are commonly used turbo machines which deliver air at a desired high velocity and mass flow rate and moderate static pressure. The pressure rise across a blower is high comparative to fan and is of the order of some millimeters of water gauge. The rise in static pressure across a blower is relatively higher and is more than 1000 mm of water gauge that is required to overcome the pressure losses of the gas during its flow through various passages.

CENTRIFUGAL BLOWER WORKING

A large number of varieties of blowers for relatively high pressure applications are of centrifugal type. The main components of a centrifugal blower are shown in Figure1. A blower consists of an impeller which has blades fixed between the inner and outer diameters. The impeller can be mounted either directly on the shaft extension of the prime mover or separately on a shaft supported between two additional bearings. Air or gas enters the impeller axially through the inlet nozzle which provides slight acceleration to the air before its entry to the impeller. The action of the impeller swings the gas from a smaller to a larger radius and delivers the gas at a high pressure and velocity to the casing. The flow from the impeller blades is collected by a spiral-shaped casing known as volute or spiral casing. The casing can further increase the static pressure of the air and it finally delivers the air to the exit of the fan. The centrifugal blower impeller can be fabricated by welding curved or almost straight metal blades to the two side wall of the rotor. The casings are made of sheet metal of different thickness and steel reinforcing ribs on the outside. Suitable sealing devices are used between the shaft and the casing. Various types of blades are used in the blowers on the basis of application. Forward curved, backward curved, radial and airfoil is the commonly used blade in the centrifugal blowers. Centrifugal blowers are used extensively in industries and HVAC applications due to their simplicity and low costs. The blowers are mainly used for air or gas handling, cooling and exhaust purpose. This type of turbo machine has small chord line length, high width and blades that are joined together with shroud and hub. This wheel type is most often called a squirrel cage wheel.

TYPES OF BLADE AND THEIR APPLICATION

There are many types of blower blades are used like forward curved, backward curved radial and airfoil. Selection of blade is based on the application. Forward curved blades are small and curved forward in the direction of the wheels rotation. These types of blower run at a relatively low speed to move the air. These types of blower are primarily used for low pressure applications, ventilating and air conditioning such as domestic furnaces, central station units and packaged air conditioning equipments. Backward inclined blades are flat and lean away from the direction of the wheels rotation. This blower runs at a relatively high speed to move a given amount of air. It is more efficient than the other types of fan. Such types of blowers are general used for heating, ventilation and air conditioning systems. Used in many industrial applications where the airfoil blade might be subjected to erosion from light dust. In the radial blade, wheel is like a paddle wheel with or without side rims. The blades are perpendicular to the direction of the wheels rotation and the blower runs at a relatively medium speed to move a given amount of air. It is used for high pressure industrial requirements.

Airfoil has the highest efficiency and runs at a slightly higher speed than the standard flat blade to move a given amount of air. It is usually used in both larger HVAC (Heating ventilation and air conditioning) system and clean air industrial applications where the energy savings are significant. It can be made with special construction for dusty air. Radial blade is also designed for material handling or dirty or erosive applications and is more efficient than the radial blade.

Basic definitions associated with blower

Static Pressure: It is the air pressure caused by its degree of compression. It can be positive or negative. In the blower it is equivalent to the difference between the static outlet pressure and the total inlet pressure.

Dynamic Pressure: It is the pressure caused by movement of air. Dynamic pressure can only be positive. In the blower it is equivalent to the average of the speeds at the outlet.

Total Pressure: It is the air pressure caused by compression and movement. It is the algebraic addition of the dynamic and static pressures at a certain point. Therefore, if the air is motionless, the total pressure equals the static pressure. In the fan, it is the difference between the total pressures determined at its outlet and inlet.

LITERATURE REVIEW

Rui Rong et al. studied the centrifugal blower with slots in the blade and cut from pressure side to suction side. Author concluded that slotted technique is effective to reduce blade surface resistance and increases lift force. That in turn save the energy and improve the characteristic of flow field. Analysis accomplished using computational fluid dynamics.

When the airfoil is working, the geometry of the channel that the fluid flows through changes with the airfoil profile. It makes the magnitude and direction of the velocity change, and resulting in flow separation. The impact produced by flow separation causes the loss of the flow's energy. In this investigation author takes the blade of G4-73No.8D centrifugal blower for the research purpose. The flow and aerodynamic characteristics of the blades under different incidence angles are analyzed and researched by means of CFD. The slotted technique on centrifugal blower blade with slots cut along the blades pressure side to suction side is adopted. And slotted blade models data are compared with generic models. The results show that the method to create a slot in blade can control the stall of centrifugal blower flow passage availably. And under the appropriate parameters, slotted model can decrease the blade surface resistance and increase lift force effectively. [2]

Kishokanna Paramasivamet al. analyzed the noise level in centrifugal blower by guide vanes. In this investigation researcher has modify the leading edge of the guide vane and found that strength of pressure fluctuation and blade passing frequency reduces. Flow pattern studies are done using CFD. Research shows the work aiming for tonal noise reduction in a centrifugal fan. In previous studies, it is well documented that tonal noise is the dominant noise source generated in centrifugal blowers. Tonal noise is generated due to the aerodynamic interaction between the rotating impeller and stationary diffuser vanes. The generation of tonal noise is related to the pressure fluctuation at the leading edge of the stationary vane. The tonal noise is periodic in time which occurs at the blade passing frequency (BPF) and its harmonics. However, omitting stationary vanes will lead to the increase of non-rotational turbulent noise resulted from the high velocity of the flow leaving the impeller. Hence in order to reduce the tonal noise and the non-rotational noise, guide vanes were designed as part of this study to replace the diffuser vanes, which were originally used in the chosen centrifugal fan. The sound pressure level at blade passing frequency (BPF) is reduced by 6.8 dB, the 2nd BPF is reduced by 4.1 dB and the 3rd BPF reduced by about 17.5 dB. The overall reduction was 0.9 dB. The centrifugal blower with tapered guide vanes radiates lower tonal noise compared to the existing diffuser vanes. These reductions are achieved without compromising the performance of the centrifugal fan. The behavior of the fluid flow was studied using computational fluid dynamics (CFD) tools and the acoustics characteristics were determined through experiments in an anechoic chamber.[3]

Lin, Sheam-Chyun, and Ming-Lun Tsai et al. Aims to provide all the technical information necessary to comprehensively evaluate the performance of the fan. A centrifugal fan with a diameter of 80 mm inclined backward is used as a demonstration object. The numerical results are used to perform a detailed flow visualization, torque calculation, efficiency estimation, and noise analysis. The results show that the fan performance curve and the SPL spectrum of the experiment are consistent with those of the numerical simulations. In addition, this study proposes two modification alternatives based on the visualization of the flow at each operative point and which confirm the successful improvement of the fan performance through numerical calculation. Therefore, this study establishes an integrated aerodynamic, acoustic and electromechanical evaluation scheme that can be used as an important tool for fan designers.

The classification of the fan power is carried out in different operating conditions. The results include flow visualization, efficiency estimation, and noise analysis. Modification alternatives are made to improve the performance of the fan. [4]

Chunxi, Li, Wang Song Ling, and Jia Yakui et al. Provide influence of the increased impeller on a spiral without changes in the performance of the centrifugal fan G4-73 is investigated. Comparisons are made between the fan with the original impeller and two larger impellers with impeller diameter increments of 5% and 10% respectively in the numerical and experimental studies. The internal properties are determined by numerical simulation, suggesting that with a larger impeller more volute loss occurs in the fan. The results of the experiment show that the flow, the increase in total pressure, the power of the shaft and the sound pressure level have increased, while the efficiency has decreased when the fan works with a larger impeller. Variation equations are suggested in the performance of the operating points for the fan with enlarged wheels. Comparisons between the results of the experiment and the clipping laws show that the clipping laws for the ordinary situation can predict the performance of the extended lower error fan wheel for a higher flow rate, although the situation of the application does not match. The noise frequency analysis shows that a higher noise level with the larger impeller fan is due to the reduction in the distance between impeller and volute.

The influence of the impeller extension is evaluated numerically and experimentally. The equations of variation of the operating points for the enlarged wheels are derived. The expansion of the impeller causes a higher noise of the fan as the distance between the impeller and the spiral is reduced. [5]

Datong, Qi, Mao Yijun, Liu Xiaoliang, and Yuan Minjian et al. investigation focused on the sound of the backward curved blades (BC) and the multi-bladed centrifugal fans forward (FC). In this article, an experimental study was conducted to investigate the noise reduction of a centrifugal industrial FC fan. First, the performance and noise characteristics of the FC centrifugal fan were tested to compare the similarities and differences with those of the BC blades and the FC multi-blade centrifugal fans. Subsequently, several different geometric configurations of the spiral were made to examine the effects of the inclined spiral tongue, the pitch of the impeller blade, the passage of the hub coil and its coupling effect on the performance and noise of the centrifugal fan FC. [6]

CFD ANALYSIS OF CENTRIFUGAL BLOWER

Fluid dynamics is the science of fluid motion. Fluid flow is commonly studied in one of three ways Experimental fluid dynamics, Theoretical fluid dynamics and numerically. In this chapter our focus was based on the CFD procedure and solver for centrifugal blower analysis.

Computational fluid dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The result of CFD analyses is relevant in, conceptual studies of new designs, detailed product development, troubleshooting, Redesign, CFD analysis complements testing and experimentation and Reduces the total effort required in the laboratory.

Procedure of CFD analysis

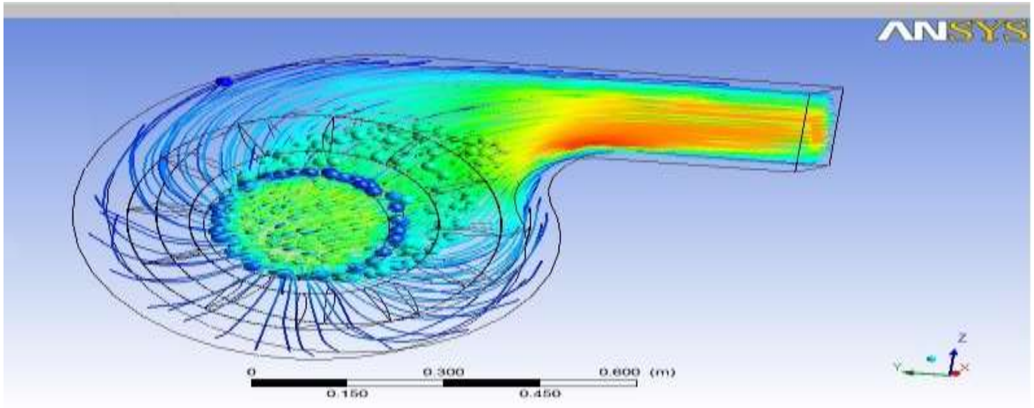
Analysis begins with a mathematical model of a physical problem. Fluid properties are modeled empirically. Simplifying assumptions are made in order to make the problem tractable (example steady-state, incompressible, non viscous). Provide appropriate boundary conditions for the problem. Discretization domain is discretized into a finite set of control volumes or cells. The discretized domain is called the grid or the mesh.General conservation (transport) equations for mass, momentum, energy, etc., are discretized into algebraic equations. The complete procedure of centrifugal blower analysis are given below:-

1. Once the design is being finalized, all components assembled to make impeller casing assembly and the same is exported from CATIA in STP format to get it Readable in ANSYS 2014 workbench.
2. After the geometry is being successfully imported, our next task is to assign meshing for which we have named the casing, inlet, outlet as well as impeller body and casing body differently.
3. Then keeping the default mesh size we have generated and update the mesh in ANSYS.
4. The setup window is now open to give the details about the boundary conditions, Solution method and to run the calculation accordingly.
5. In the present analysis we have taken energy model to solve along with the fluid And solid medium as air (Casing) and aluminum (Impeller).
6. A simple velocity pressure coupling method is adopted along with a least square cell based gradient and second order upwind type momentum.
7. Now we have initialized the solution by choosing hybrid initialization method.
8. The calculation is being run by taking number of iteration as 3.
9. Now in visualization workbench we have plotted contour for pressure variation.
10. Now the results are being compared with numerical one to get comparative study.

Pressure contour for different flow rate is plotted in ANSYS14. Results are following below. Figure4.1 shows the pressure contour of centrifugal blower with blade of maximum thickness of 5mm. In this chapter we will describe the flow pattern and pressure variation in the centrifugal blower which is obtained by colourful fluid dynamics. In this state the air of density 1.2 kg/m3flow at inlet with the rate flow rate of 0.1m3/s and the rotation of impeller is 1500 RPM.

Air flow at the inlet is just below the atmospheric pressure due to the rotation of the impeller negative pressure is created at the blower inlet. And when air strikes on the blade, its pressure continuously increases and velocity decreases. We have set the pressure scale in Pascal. As we can see from the figure1 that maximum intensity of pressure is occurs at the blower outlet which is in the magnitude of 1.583m in form of head, which denotes the average pressure at blower outlet. It is the beauty of the colorful fluid dynamics that we can see the pressure variation easily in the figure and red color at blower exit duct shows the higher outlet pressure.

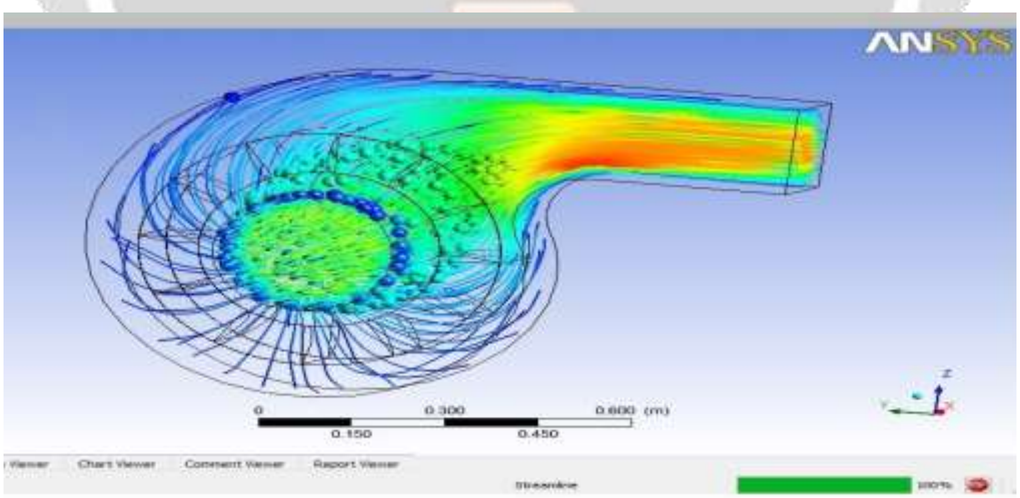
Analysis of centrifugal blower with varying outlet diameter:



Pressure contour of centrifugal blower at air flow rate 0.1 m^3/s

Represent the pressure contour of centrifugal blower with same airfoil blade which was analyzed earlier in the same virtual environment, the model is being tested for the same working condition except the air flow rate which is being increased to 0.2 m^3/s. We will describe the flow pattern and pressure variation in the centrifugal blower which is obtained by computational fluid dynamics. In this state the air of density 1.2 kg/m^3flow at inlet at the rate of 0.2 m^3/s and the rotation of impeller is being constant which is taken as 1500 RPM. Air flow at the inlet is just below the atmospheric pressure due to the rotation of the impeller negative pressure is created at the blower inlet. And when air strikes on the airfoil blade, its pressure continuously increases. Here again we have set the pressure scale in Pascal. As we can see from the figure4.2 that maximum intensity of pressure is occurs at the blower outlet which is in the magnitude of 4.933m in terms of head, which denotes the average pressure at blower outlet.

It is the beauty of the computational fluid dynamics that we can see the maximum value of pressure easily in the figure and red colour at blower exit duct shows the higher outlet pressure and blue colour represent the minimum pressure. Here we can easily see that with increasing flow rate of air, head decreases.



Pressure contours of centrifugal blower at air flow rate 0.2m^3/s.

At an angle of 60^0on impeller periphery and enclosed by casing. We have do good try to represent the flow pattern and pressure variation in the centrifugal blowerwhich is obtained by CFD. Air of density 1.2 kg/m^3flow at inlet with the flow rate of 0.3 m^3/s and again the impeller speed is being constant which is taken as 1500 RPM. We have set the pressure scale in Pascal. As we can see from the above figureure8 that maximum intensity of pressure is occurs at the bloweroutlet which is in the magnitude of 22.26m in terms of head, which denotes the average pressure at bloweroutlet.

References:

[1] Yahya, S. M. Turbines compressors and fans. Tata McGraw-Hill Education, 1987.

[2] Rong, Rui, Ke Cui, Zijun Li, and Zhengren Wu. "Numerical Study of Centrifugal Blowerwith Slots in Blade Surface." Procedia Engineering 126 (2015): 588-591.

[3] Paramasivam, Kishokanna, Srithar Rajoo, and Alessandro Romagnoli. "Suppression of tonal noise in a centrifugal blowerusing guide vanes." Journal of Sound and Vibration 357 (2015): 95-106.

[4] Lin, Sheam-Chyun, and Ming-Lun Tsai. "An integrated performance analysis for a backward-inclined centrifugal fan." Computers & fluids 56 (2012): 24-38.

[5] Chunxi, Li, Wang Song Ling, and Jia Yakui. "The performance of a centrifugal blowerwith enlarged impeller." Energy Conversion and Management 52, no. 8 (2011): 2902-2910.

[6] Datong, Qi, Mao Yijun, Liu Xiaoliang, and Yuan Minjian. "Experimental study on the noise reduction of an industrial forward-curved blades centrifugal fan." Applied Acoustics 70, no. 8 (2009): 1041-1050.

[7] Chintala, Gopinath, and Prasad Gudimetla. "Optimum material evaluation for gas turbine blade using Reverse Engineering (RE) and FEA." Procedia Engineering 97 (2014): 1332-1340.

[8] Worasinchai, Supakit, Grant Ingram, and Robert Dominy. "Unsteady surface pressuresand airload of a pitching airfoil." Energy Procedia 34 (2013): 407-416.

[9] Danmei Hu, Jia Li, Haijin Yan.Numerical Simulation of Airfoil Dynamic Stall of Horizontal Axis Wind Turbine[J]. Proceedings of the CSEE, 2010,30(20):106

[10] Ming Guo, Xiaoyu Zhen, Sheng Zhou. BVF application on blowing control of flow separation of a compressor cascade[J]. Journal of Aerospace Power, 2008˙23(8)˖1498-1503.

[11] Mohamed Hafez6̑Essam Wahba□Simulations of viscous transonic flows over lifting airfoils and wings[J]□Computers & Fluids, 2007, 36: 39- 52.

[12] Patil, Sunil R., et al. "Effect of Volute Tongue Clearance Variation on Performance of Centrifugal Blower by Numerical and Experimental Analysis." Materials Today: Proceedings 5.2 (2018): 3883-3894.

[13] Charapale, Utkarsh Diliprao, and Arun Tom Mathew. "Prediction of Flow in the Industrial Blower using Computational Fluid Dynamics." Materials Today: Proceedings5.5 (2018): 12311-12319.

[14] Hering, Martin, Stefanie Wahl, and René Meise. "Characteristic blower map modeling for volume flow rate estimation in solid oxide fuel cell systems." Flow Measurement and Instrumentation (2018).