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 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. Centrifugal blowers are used extensively in industries and heating, ventilation and air conditioning 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. Due to versatility of use of blower, a detailed research work is still needed in this direction.

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 millimetres 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. [1] Figure1.1 shows the main components of centrifugal fan.

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.

The design of blower blades constitutes the most significant feature of the blowers. The blade shape, which is of aerofoil sections in the case of centrifugal blowers, plays an important role in the performance of the blowers and blower efficiency is greatly dependent on the blade profile. Therefore, this task of improving energy efficiency can be accomplished by reducing losses by reducing the flow recirculation and flow separation in the blowers by using aerodynamically designed blower blades made of suitable material. To meet the mentioned task a suitable aerofoil section is very much required.

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 blowerit 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

Patil, Sunil R et al., focuses on the influence of the volatility of the tongue clearance on the performance of the backward curved blades. The four types of centrifugal fan housings with different tongue diameters of 6%, 8%, 10% and 12.5% of impeller diameter were used for numerical and experimental analyzes. The computational fluid dynamics model was developed for numerical analysis based on the experimental configuration. The Navier-Stokes equations averaged by Reynolds with the standard turbulence model k-ε were discretized using finite volume approximations. The numerical results were validated with the experimental results using IS 4894-1987. Performance parameters were calculated, such as the total pressure, efficiency and flow rate of the blower. The results show that the distance of the volume gap has a significant impact on the performance of the centrifugal blower and that these parameters increase when the tone of the volute decreases. [12]

Charapale, Utkarsh Diliprao, and Arun Tom Mathew et al.,suggest that the blower is one of the key components in all industrial applications such as boilers, cooling systems, dryers, etc. Therefore, the performance and characteristics of the blower affect the overall system and the efficiency of the work. The blower system consists mainly of three parts: suction cup, impeller, and spiral, which determine the character of the blower. It is very important to predict the airflow for a fan before use. The Industrial Centrifugal Blower is conducting a study to accurately predict airflow and mass flow using Computational Fluid Dynamics (CFD) tool. The detailed blower model is created using 3D CAD software and CFD code. Fluent 3D solves various control equations and predicts the flow at the fan output. The numerical results obtained using CFD are compared with the characteristic provided by the manufacturer. [13]

Hering, Martin, Stefanie Wahl, and René Meise et al.,model the characteristic fan based on data from the manufacturer, Application of the blower law laws for the representation of standard conditions, Replace volume flow sensor, increasing the efficiency of solid oxide fuel cell systems. In this paper, characteristic fan card modeling approaches are implemented in a solid oxide fuel cell system as a replacement for flow sensors that reduce pressure drop and investment costs. Likelihood similarity laws and various regression modeling approaches based on detailed manufacturer information are used to express volumetric flow as a function of measured fan speed, pressure, and temperature. The indirect model-based volume flow calculation is compared to a direct volume flow measurement in a test bench using an ambient air side channel blower. In fan modes with low to medium pressure differentials, a high degree of correspondence between indirect and direct volumetric flow measurement is achieved, demonstrating the high accuracy of the model. In addition, the superiority of the modeling approach is underlined by a lower absolute margin of error and an average error margin of 55% and 66%, respectively. Replacing the air flow rate sensor with a characteristic blower model modeling to determine airflow in a 10kW solid oxide fuel cell system results in potential increases in electrical efficiency between 0.5% and 0.7% due to the reduction in pressure drop. [14]

Hariharan, C., and M. Govardhan et al.,made an attempt to explore the energy-efficient scroll for industrial blowers, Parallel wall volutes for aerodynamic and rectangular performance, The parallel wall spiral R 4.0 achieves a higher efficiency of 6%, Parallel murals can be an efficient and effective alternative. This article proposes parallel wall spirals as an alternative and energy efficient spiral to the rectangular spiral commonly used for an industrial centrifugal fan. A detailed performance comparison is made between the parallel wall and the rectangular spiral for four different aspect ratios. The analysis of performance in the scenario suggests that the volutes of the parallel wall perform better in terms of specific workload and overall isentropic performance. By evaluating the performance of the components, it has been found that the parallel wall spirals have a greater recovery of static pressure and less loss for the entire operating range considered. The flow field shows that the parallel wall spiral has a more uniform static pressure distribution at the entrance of the spiral compared to the rectangular spiral. A detailed analysis of the aerodynamic performance shows that the overall performance of a centrifugal blower with a parallel wall spiral can be improved by up to 6%. In addition, it is shown that the rectangular spiral can be replaced by a parallel wall spiral without changing the base, the inlet, and the outlet channel. [15]

Lee, Young-Tae, and Hee-Chang Limet al.,have carried out an optimized design of a centrifugal fan with different fan ribs, We investigate the effects of the internal components of the fan on the pressure fluctuations of the surface pressure, measurements are carefully designed to achieve a reasonable trend, The calculations are consistent with the experiment, The curved fins forward show the best performance. This study aims to develop an optimized design of a centrifugal fan consisting of different fan ribs based on performance ratings after changes in the shape of its internal components. Several components, such as the outer casings and the rotating fan ribs, which are arranged in different operating conditions, are evaluated numerically and experimentally. Classification is based on performance parameters, including an inlet and outlet pressure, flow, torque, and radial fan power. The numerical analysis suggests that the combination of the multiple rotation frame method and the standard turbulence model for k-ε was adequate for the simulation of internal flow characteristics and yield prediction. The numerical results were compared with tests under carefully designed experimental conditions. Depending on the results and depending on the output of the fan, the flow increased gradually to 7% more than the existing model. The results of the experimental and numerical calculation were in good agreement, especially in the conditions of the initial limits to atmospheric pressure. Among the four different wheels studied, the fan type was associated with forward-curved fan fins of better performance, achieving a maximum flow of 2.2 m3 / min and a torque of 0.09 Nm. [16]

Xu, Chen, and Yijun Maoet al.,presents an experimental research on metallic foam to control the noise of a centrifugal fan. Nine metal foam specimens with different cell types are used, ie open, semi-open and closed, to compare their effects on the aerodynamic performance and noise level of the centrifugal fan. The experimental data confirm that the open-cell metal foam is most effective in controlling fan noise, as it not only significantly suppresses tonal noise, but also attenuates broadband noise. In addition, the geometric parameters of open-cell metal foams, ie pores per inch and porosity, are investigated to investigate their effects on the aerodynamic performance and noise level of the centrifugal fan. [17]

Baloni, Beena D., Yogesh Pathak, and S. A.et al.,Channiwala validated simulation model is chosen for the optimization of the spiral according to the Taguchi method and the ANOVA, Experimental analyzes indicate that the optimized spiral works better than the original. To the general view of the parameters. Reducing the percentage of change in the static pressure at the outlet of the impeller, minimizing the losses within the displacement and maximizing the back pressure at the displacement output is a good selection of activities. In this method, the numerical simulation of the 3-D flow in a one-stage centrifugal fan coil is carried out using the FLUENT software for matrix experiments. These matrix experiments are proposed by the Minitab software. The simulation case is carried out using flow, turbulence and energy equations with the SIMPLE print speed coupling. The experimental performance of the optimized configuration is performed. The result shows better performance with an optimized spiral. It is observed that an optimized centrifugal fan has 7.4% more efficiency.[18]

METHODOLOGY

System description & Flow diagram

The complete procedure of blower analysis. Task was initiated by generation of two dimensional blade profile by using x and y coordinate. Generated profile is extruded up to the required width and in the next stage we mount it on the impeller. Now casing assembly is done with impeller using concentricity command. We assign the material of impeller as Aluminium. Solid modelling of centrifugal blower is completed at this stage. To fulfil the aforesaid purpose ANSYS14.0 workbench is used in the configuration of 64 bit processor and 8 GB RAM. Geometry is now being imported in the ANSYS FLUENT workbench. Now we define the boundary condition as pressure inlet, velocity inlet and set z- coordinate as a rotational axis. Now update the geometry and initiate the solution. Obtained result from CFD is compared with analytical solution.

Procedure

Complete procedure of blower analysis is given below.

1. Task was initiated by generation of two dimensional blade profile by using x and y coordinate.
2. Generated profile is extruded to the required width and in the next stage we mount it on the impeller.
3. Now casing assembly is done with impeller using concentricity command.
4. Assign the material of impeller as Aluminium. Solid modelling of centrifugal blower is completed at this stage.
5. To fulfil the aforesaid purpose ANSYS14.0 workbench is used.
6. Geometry is now being imported in the ANSYS FLUENT workbench. Now we define the boundary condition as pressure outlet, velocity inlet and set z- coordinate as a rotational axis.
7. Now update the geometry and initiate the solution. Obtained result from CFD is compared with analytical solution.
8. Same procedure is being repeated for different blade angles.

Analysis process :-

1. CAD model : CAD model is first prepared in CREO 2.0 and is converted in IGES format to export it to ANSYS CFX
2. The Computational Domain: The appropriate size and shape of the computational domain, also referred to as control volume, and the best placement of the model in the domain, needs to be determined. A domain too large will make the simulation unnecessarily large and waste computational resources, however a domain too small will lower the accuracy of the results. The properties of the domain such as temperature, pressure and fluid properties need to be chosen.
3. Boundary Conditions: The conditions at the boundary of the domain need to be set such as inlet velocities, outlets and wall attributes.
4. Discretization of the Domain: Since CFD utilizes numerical solutions the domain needs to be discretised or meshed as it is more commonly referred to. The mesh will have to be refined in areas with high gradients

for example close to the surface around the aircraft model. Initial Values Initial values need to be set for all the nodes in the domain.

5. Convergence Criteria: The Criteria for which the simulation can be regarded as converged needs to be determined.
6. Post Processing: The simulation results will give the sums of the forces acting in each direction on the model, or any chosen part of it. Through this, the drag coefficient CD can be obtained. The amount of lift created by the wing will have to be taken into account in order to be able to calculate the induced drag. Subtracting the induced drag from the total drag should give the zero lift drag.

RESULTS AND DISCUSSION

Centrifugal blower was successfully analysed numerically and computationally. Pressure distribution and head at exit duct was calculated for different flow rate of air and compare with computational results. Table represent the pressure head and mass flow rate with varying outside diameter of impeller for centrifugal blower.

Sr. No.	Outside diameter	Mass flow rate in kg/s	Pressure head in m	Pressure head in m using CFD	Percentage deviation
1	350	0.12	1.58	1.76	12.82
2	400	0.24	4.93	5.46	10.75
3	450	0.36	7.42	7.95	7.14
4	500	0.48	9.89	10.41	5.25
5	550	0.60	11.95	12.39	3.66

It can be easily seen that with increase in mass flow rate pressure head continuous increases numerically as well as computationally.

.

Variation in pressure head with different blade angles

Sr. No.	Blade angles in degree	Mass flow rate in kg/s	Pressure head in m	Pressure head in m using CFD	Percentage deviation
1	30	0.12	1.58	1.76	11.39
2	45	0.24	7.47	8.12	8.7
3	60	0.36	22.26	23.01	3.36
4	75	0.48	63.11	63.61	0.79
5	85	0.60	293.10	276	6.15

It can be easily seen that with increase in flow rate pressure head continuous goes to decreases numerically as well as computationally. Finally it has to be found that in first case blade are investigated with variation in impeller diameter and in second case investigation was completely focused on blade angles

CONCLUSIONS

In this research work the effect of two different parameters namely blade angle and impeller diameter of centrifugal blower were investigated. The results obtained from CFD analysis and numerical approach are correlated successfully. It was found that flow separation and recirculation can be clearly observed in the CFD analysis. The blade profile plays a significant role to reduce flow separation occurs at trailing edge of the blower.

Ifthe blades of centrifugal blowers are properly inclined it can provide better performance to the blowers and help in minimizing energy consumption. This analysis helps in proper selection of blower for variety of applications.

Pressure distribution was observed in terms of head and visualize in the centrifugal blower while considering the optimizing parameter stated above. And compare the results, obtained from numerical approach. In this work we have used two different centrifugal blowers and analyzed by K-epsilon model. Finally CFD tool is best suitable to meet our demand.

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 Hafez6͡Essam 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).

[15] Hariharan, C., and M. Govardhan. "Improving performance of an industrial centrifugal blower with parallel wall volutes." Applied Thermal Engineering 109 (2016): 53-64.

[16] Lee, Young-Tae, and Hee-Chang Lim. "Performance assessment of various fan ribs inside a centrifugal blower." Energy 94 (2016): 609-622.

[17] Xu, Chen, and Yijun Mao. "Experimental investigation of metal foam for controlling centrifugal fan noise." Applied Acoustics104 (2016): 182-192.

[18] Baloni, Beena D., Yogesh Pathak, and S. A. Channiwala. "Centrifugal blower volute optimization based on Taguchi method." Computers & Fluids 112 (2015)： 72-78