EXPERIMENT & CFD ANALYSIS OF BLADE OF DUST COLLECTOR & ITS OPTIMIZATION

[1]Raj D. Patel, [2]Mr. Pravin Zinzala,

[1]PG Student, [2]Asst. Professor
[1] Thermal engineering,[2] Mechanical Engineering Department
[1, 2]LJInstituteofEngineeringandTechnology, Ahmedabad,Gujarat, India
[1]rajdpatel45@gmail.com, [2]zinzalapravin@gmail.com

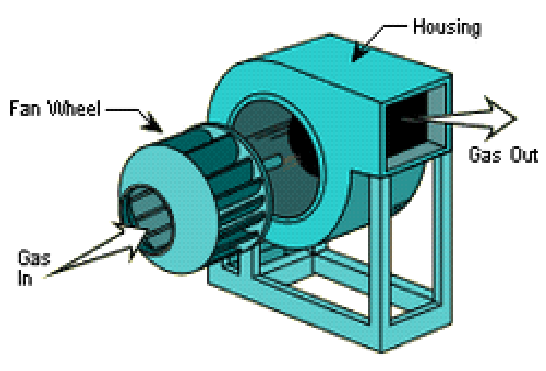
Abstract In this thesis, effect of geometric parameters of a centrifugal blower with backward- and forward-curved blades has been investigated. Centrifugal Blowers are used for enhancing the heat dissipation from the cutting tools. As a first step, an experimental setup was developed and prototypes of Blowers were made to carry out measurements of flow and power consumed by the Blower. The Blower mounting setup was such that Blowers with uniform blades can be tested. Next, a computational fluid dynamics (CFD) model was developed for the above setup and the results are validated with the experimental measurement. Further, parametric studies were carried out to quantify the power coefficient, flow coefficient, efficiency and flow coefficients.

IndexTerms – blower unit, Inlet angle & outlet angle of impeller ,CFD analysis, DOE Taguchi method

I. INTRODUCTION

A centrifugal fan is a mechanical device, which is used for moving air or other gases. The terms "blower" and "squirrel cage fan" because it looks like a hamster wheel is also used as synonyms. These fans escalate the speed of air stream with the rotating impellers. They use the kinetic energy of the impellers or the rotating blade to increase the pressure of the air/gas stream which in turn moves them against the resistance due to ducts, dampers and other components. Centrifugal fans accelerate air radially, changing the direction (typically by 90°) of the airflow. They are study, quiet, reliable, and capable of operating over a wide range of conditions.

This paper presents a computational study of the blower unit and how to increase the performance using computational fluid dynamics. The main aim of blower is to deliver the gas or air continuously with an allowable rise in pressure to overcome resistance in the flow and they can achieve high pressures than fans. They are also used to produce negative pressures for industrial systems. Centrifugal blowers are constant volume devices at constant speed. Usually in many industries, blowers are used in dust collection system. These blowers suck the air from the inlet and discharge it through the outlets for cleaning the machines. It is power consuming machine in which mechanical work is converted into the pressure head of air or gas. Electric motors are used as power source.Centrifugal fan is a constant CFM device also called a constant volume device, meaning that, at a constant fan speed, a centrifugal fan will send a constant volume of air rather than a constant mass. This means that the air velocity in a system is constant even though mass flow rate through the fan is not.The centrifugal fan is one of the most used fans. Centrifugal fans are by far the most prevalent type of fan used in the HVAC industry today. They are usually cheaper than axial fans and simpler in construction. In automotive industries, fans are used for cooling purpose in IC engines. The fan get its energy from the power generated by the engine. It is used in transporting gas or materials and in ventilation system for buildings. They are also used commonly in central heating/cooling systems. They are also well-suited for industrial processes and air pollution control systems. When one fan can't perform the necessary flow and pressure, the fans must be run in series two or more, in order to achieve the required pressure and flow. The pressure of the centrifugal fan is high, so it is widely used in the production and has high using frequency, the centrifugal fans in series are often used in practical production. From the relevant statistics, fan power consumption accounting for 12% of the total electricity consumption.



.Fig 1 schematic Diagramme of blower unit

II. LITERATURE SURVEY

AtrePranav C. and ThundilKaruppa Raj R [1] “Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers ”.There are mainly three parameters static pressure, static efficiency and 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. The Numerically obtained volute casing design has drawback of recirculation phenomenon which was clearly observed in CFD analysis. The cut off height plays the major role in prevention of recirculation occurring near outlet region of the fan system. The k-ω (2 eqn. SST) turbulence model gives more accurate computational solution for the fluid physics.Finally, it can be concluded that 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.

Sheam-Chyun Lin, Ming-Lun Tsai,2012 [2] "An integrated performance analysis for a backward-inclined centrifugal fan” Recently, the requirement for developing high-performance centrifugal fans exists due to increasing system resistance and space limitation on computer devices. Also, performance evaluation of fan design under different operating conditions is evidently in great demand for practical engineering applications.
Therefore, this comprehensive investigation is aimed at offering the overall technical information for thoroughly evaluating fan performance. An 80 mm-diameter backward-inclined centrifugal fan is chosen to serve as the research subject for demonstration purposes. The results indicate that the fan performance curve and the sound pressure level (SPL) spectrum of the experiment agree with those of numerical simulations. In addition, this study proposes two modification alternatives based on the flow visualization at each operating point, having verified the successful enhancement of fan performance via numerical calculation. Consequently, this study establishes an integrated aerodynamic, acoustic, and electro-mechanical evaluation scheme that can be used as an essential tool for fan designers.

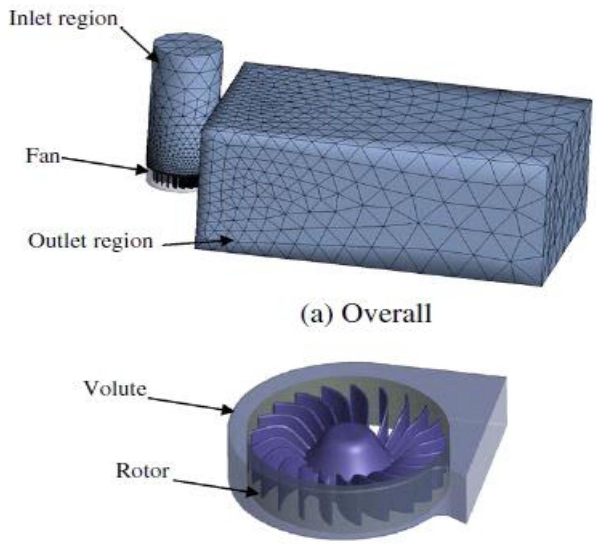


Figure: 2. Blower unit

Generally speaking, this integrated performance evaluation consists of aerodynamic performance analysis, efficiency estimation, motor characteristic identification, and noise prediction. Through this evaluation procedure, the fan designer can visualize the flow patterns and calculate the loading torque at various operating points.Additionally, the predicted fan noise qualitatively agrees well with experimental measurement at maximum volumetric flow rate. Thus fan designer can utilize the noise simulation as an analysis tool to obtain the noise fluctuation information and evaluate the fan acoustic behavior in a thorough manner. Furthermore, with the overall understanding of flow mechanism inside the centrifugal fan at various operating conditions, several modification alternatives are proposed in this comprehensive investigation to successfully enhance the fan performance. In summary, this study establishes an integrated aerodynamic, acoustic, and electro- mechanical evaluation approach that can be used as an important tool for fan designers.

N. Vibhakar #1, S. D. Masutage#2, S. A. Channiwala [3] “Three Dimensional CFD Analysis of Backward Curved Radial Tipped Blade Centrifugal Fan Designed as per Unified Methodology with Varying Number of Blades”

1. The results of numerical analysis, clearly follows the standard performance curves of a centrifugal fan. Present numerical analysis is closer to design point parameters, for centrifugal fan under study.
2. The mean pressure distribution around the volute casing is not uniform and jet and wakes are observed in the vicinity of tongue region. Since the flow is considered incompressible, pressure fluctuations at impeller outlet region is affecting flow around impeller zone.
3. The flow phenomenon of recirculation near tongue region is confirmed by numerical analysis.
4. The exact amount of circulatory flow depends upon the shape of the blade passage. For a given impeller more vanes make the passage narrower giving greater guidance to the fluid and reducing the circulatory flow effect. Hence all the quantities varying with flow coefficient are increased as the number of blades increases.
5. The streamlines clearly shows the rotating effect of blades on flow, within and outside of impeller zone, which is imposed using MRF approach.
6. 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.

Yogesh R. Pathak, Beena D. Baloni, Dr.S.A.Channiwala [4] "Numerical Simulation of Centrifugal Blower using CFX." In this paper, The Numerical Analysis has been done to understand the effect of flow coefficient on the centrifugal blower. From the results of the Numerical analysis, it can be deduced that the pressure coefficient decreases with the increase in flow coefficient.

The Flow Analysis for different mass flow rate shows that static pressure and total pressure has been increase with the decrease in flow coefficient and velocity is increased with the increase in flow coefficient.

Detailed Flow Analysis for experimental value of flow coefficient = 0.2563 is also carried out at different angular positions of volute and axial positions which shows that as we move along the volute from suction to exit, static and total pressure increases and velocity is reduces. So it can be concluded that the volute serves its function of pressure recovery.

Wan-Ho Jeon [5] "A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan" The noise of blower and fan mainly consists of vibration-induced noise and flow-induced noise. In this paper, only the flow-induced noise is under consideration. By using a numerical prediction method for the centrifugal fan noise, the characteristics of acoustic pressure from the centrifugal impeller and wedge system were studied. Numerical results and the measured data showed a consistency. The effects of design parameters, such as the rotating velocity, the cut-off distance and the number of blades of the impeller, on the noise of the fan were investigated. It was found that the most important factor for the noise of the centrifugal fan, is the cut-off distance. The cut-off distance changes the sound pressure levels not only at BPF but also at higher harmonics. As the cut-off distance increases, the amplitudes at higher harmonic frequencies decrease very steeply. It was also observed that the number of blades does not affect the noise level significantly.

D.V. Bhope, P. M. Padol [6] "experimental and theoretical analysis of stresses noise and flow in centrifugal fan impeller."

Stesses analysis of fan impeller by experimental and finite element method has shown that , the stress pattern in impeller components is highly complex. The stresses in impeller components can be reduced by using the stiffening rings on blades.

It observed that by using 2 mm thick cylindrical shaped stiffening ring has reduce stress in impeller components more than 50% as compared with the impeller without stiffening ring.

III. MODELING AND CFD ANALYSIS OF BLOWER UNIT

The modeling has been performed on the Solid works 2009 version and then after the analysis work has been performed on the ANSYS 12.0 version.For analysis we made cavity model in the solid works. In cavity model we used companies dimension and made the model according that. We modified few parameters like inlet and outlet angle of impeller to make blower unit more effective and efficient.

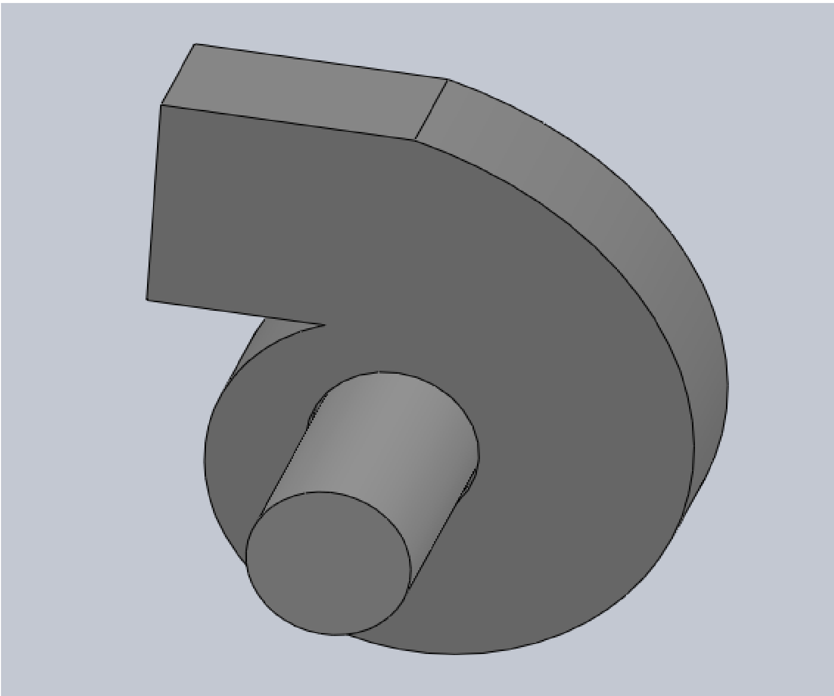


Fig. 3 Cavity model of Blower unit

- **CREATING MESH TO THE GEOMETRY**

Grid generation is very important before starting CFD calculations.The mesh created in ANSYS is intended for use influent,so it must be a single block ,structured mesh. However, this mesh can also be used in any of the other fluent solvers.This type of mesh is some times called a mapped mesh ,because each grid point has a unique i, j ,k index ..The fig 4 shows Meshof centrifugal fan. In this mesh we discretize the model in 139745 nodes and 441571 tetrahedral elements.

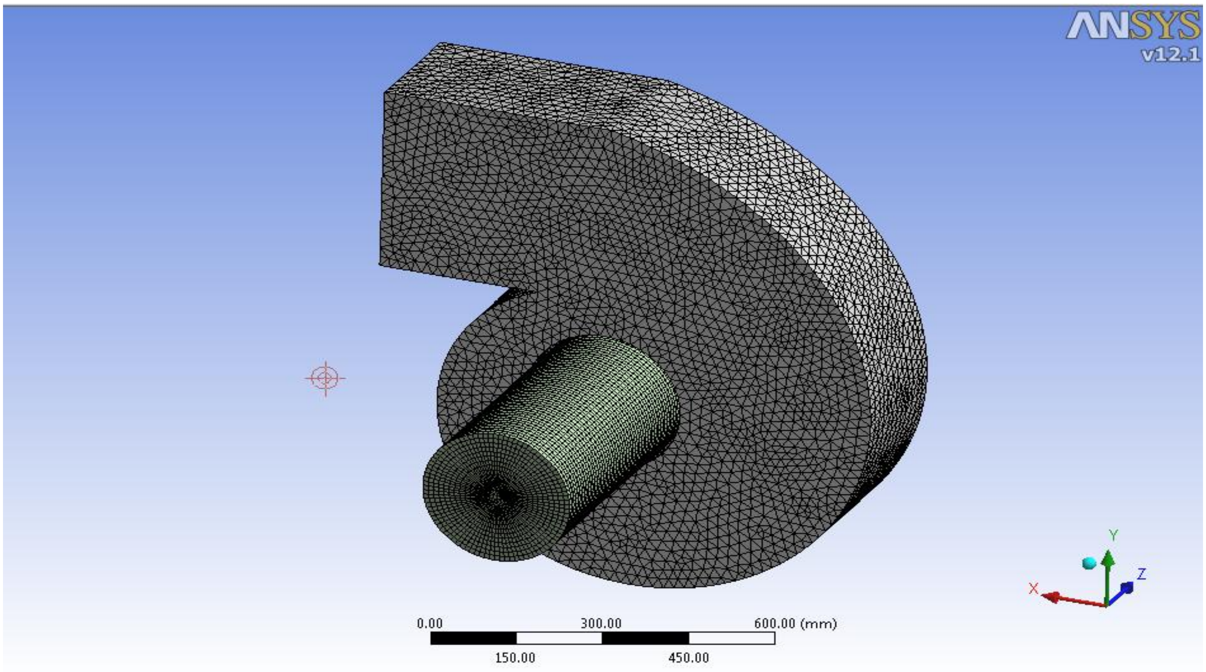


Fig. 4 Mesh of Centrifugal fan

- **Boundary Condition**

Boundary conditions applied to the different parts of the geometry. Parts are created during the process of meshing in the Ansys CFX 12.1. First boundary condition has been apply at inlet of blower unit. Second boundary condition has been apply at outlet of blower unit.

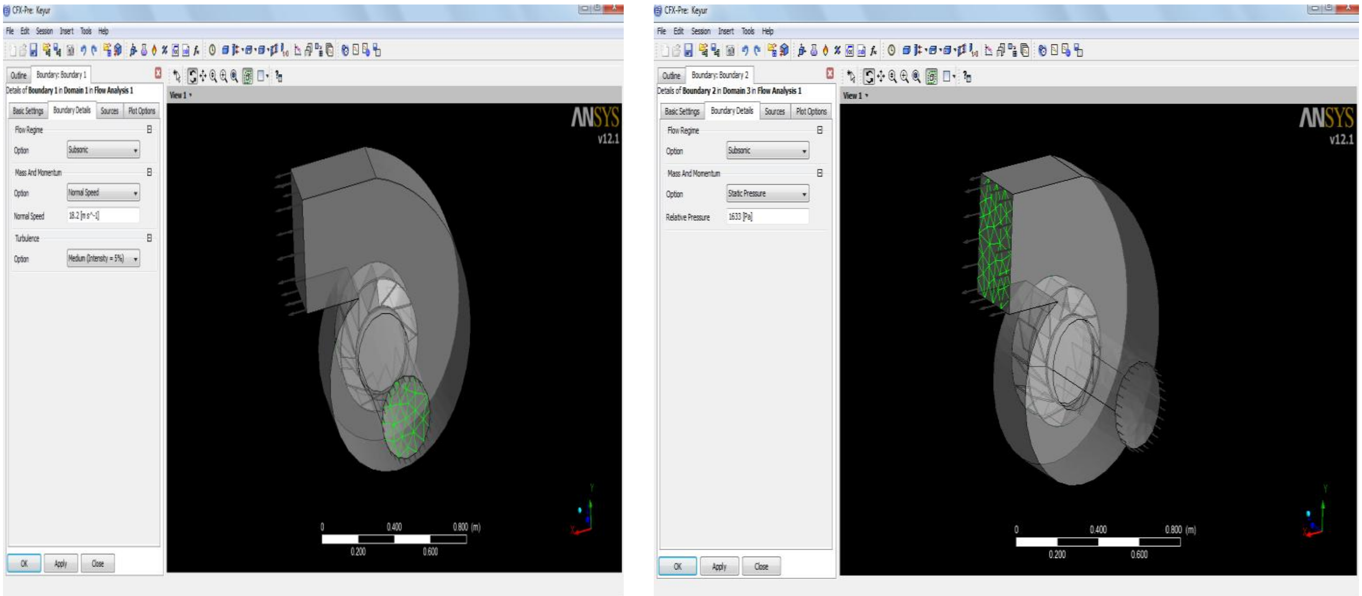


Fig. 5 Boundary Condition

Now for increasing CFM and mass flow rate we take eight different cases. In every cases we changed parameters. For this analysis we take two parameters. Inlet angle, and outlet angle. fig 6 shows screen shots of analysis.

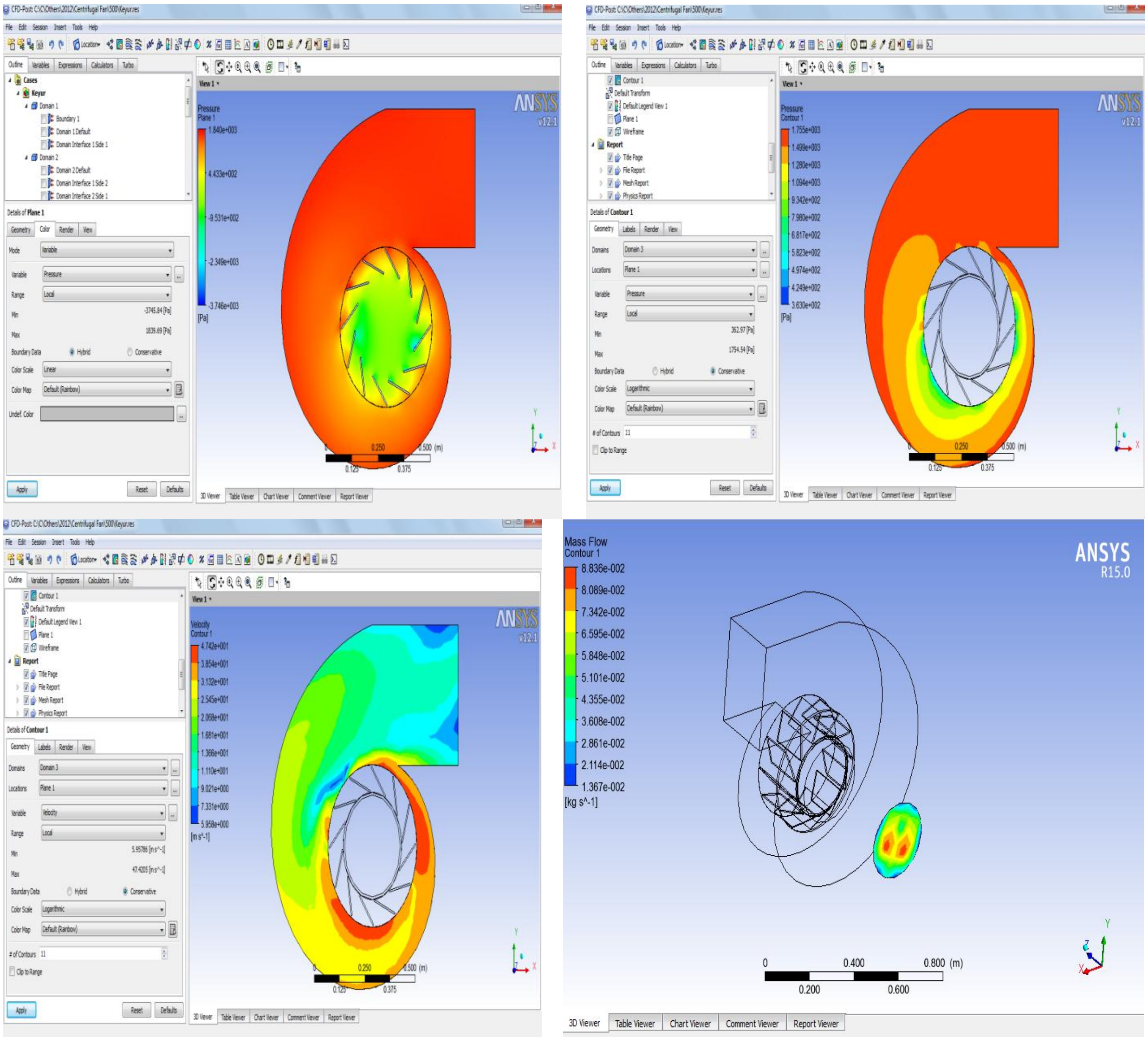


Fig. 6 Screen Shots Of Analysis

After analysis we found that if we use inlet angle 24.5 and outlet angle 18.5 then we get higher pressure as well as mass flow rate. Fig. 7 shows in Ansys cfx 12.1 and Table shows the result table.

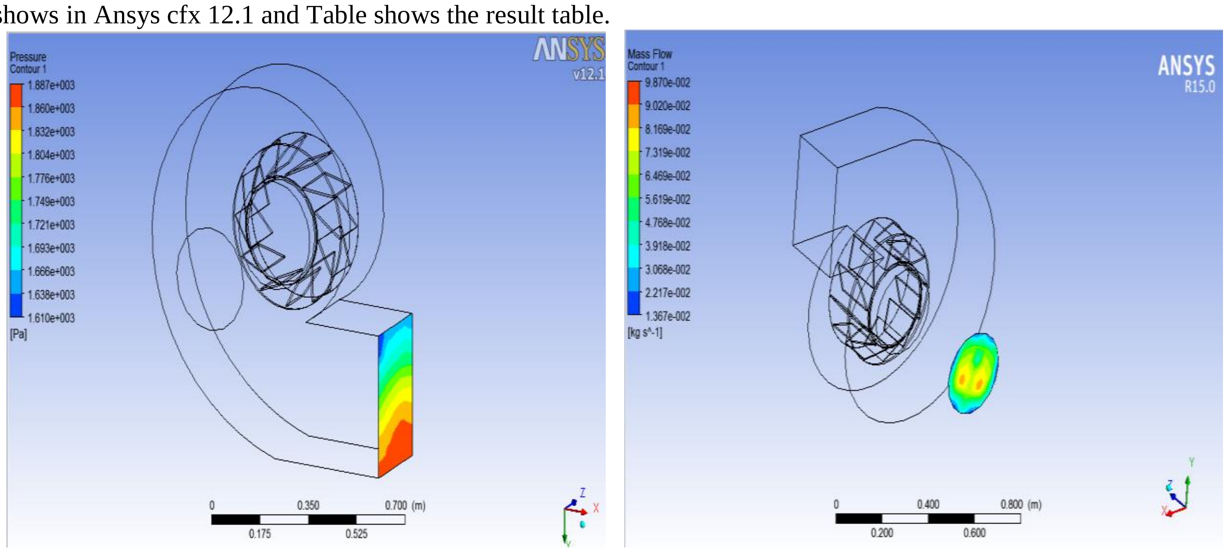


Fig.7 pressure and mass flow rate in cfd analysis

.

	Outlet Blade Angle	Inlet Blade Angle	Outlet Pressure(Pa)	Outlet Mass Flow Rate (Kg/s)
Fan 1	24.5	18.5	1887	0.0987
Fan 2	24.5	20.5	1803	0.0928
Fan 3	24.5	22.5	1720	0.0845
Fan 4	26.5	18.5	1640	0.0814
Fan 5	26.5	22.5	1484	0.0714
Fan 6	28.5	18.5	1409	0.0689
Fan 7	28.5	20.5	1312	0.0612
Fan 8	28.5	22.5	1202	0.0601

Above results shows that by changing inlet angle and outlet angle of impeller the mass flow rate and CFM changed. So that after optimization we select best parameters which gives less CO_2 emissions at outlet. For that purpose we has been optimize this value using DOE-Taguchi L9 method. Fig 8 shows result of optimization.

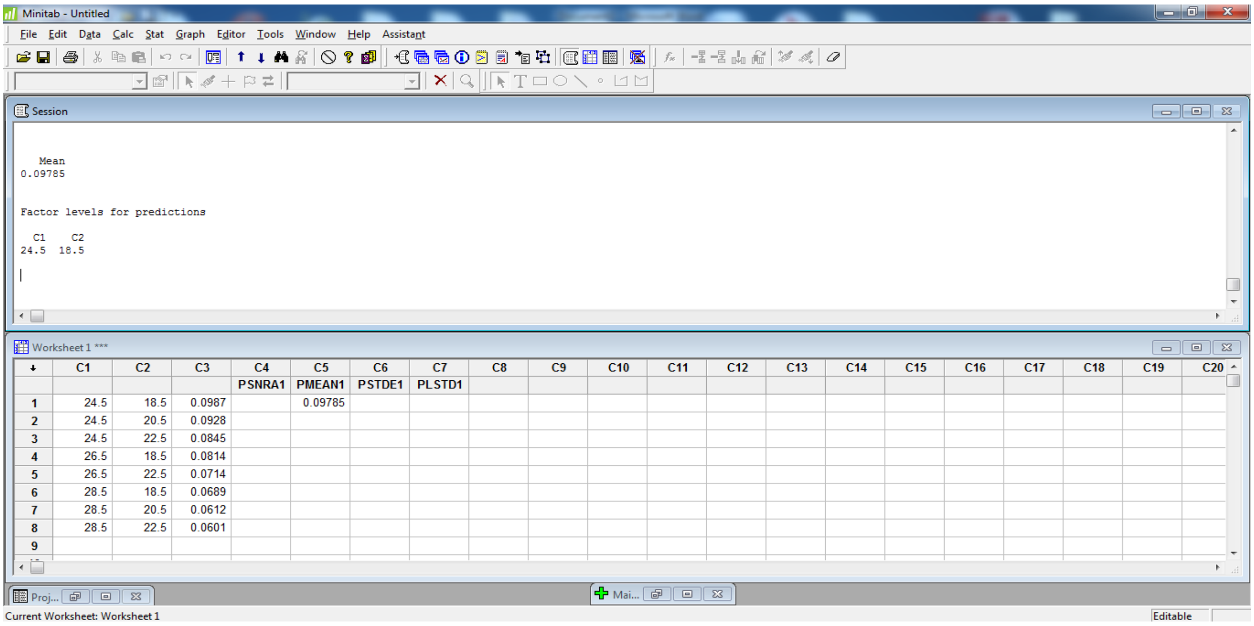


Fig 8 Result of optimization.

Using DOE Taguchi method we find the optimum values of our parameters. As per this optimization optimum value of inlet angle is 24.5°, outlet angle is 18.5°. So by using this optimum values we has been change the existing design and in above analysis c1, c2 indicates inlet angle (°), outlet angle (°) respectively.

IV. CONCLUSIONS

Blower unit is analyzed using CFD simulation in ANSYS CFX 12.1.The cavity model was discretize using unstructured tetrahedral grids. The simulation was carried out using three parameters with k – e turbulence model. The simulated results was optimized using DOE – Taguchi L9 method in Minitab. After getting Optimize values following conclusions are drawn

- Blower unit is analyzed using CFD simulation in ANSYS CFX 12.1.The cavity model was discretize using unstructured tetrahedral grids. The simulation was carried out using three parameters with k – e turbulence model.
- The simulated results was optimized using DOE – Taguchi L9 method in Minitab. Optimize values then validate the operating characteristics are predicted by the CFD. CFD simulation has been compared with the existing data.
- The optimum value of inlet angle is 24.5 band outlet angle is 18.5 with which we optimized design we able to increase the output pressure and mass flow rate.
- So that we can conclude that our analysis has been increase the amout of dust collected So that we had improves the performance and effectiveness of blower unit.

V. REFERENCES

[1] [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 science, page no: 7-11, ISSN 2277-2501, October 2012.

[2] Sheam-Chyun Lin and Chia-Lieh Huang, "An integrated experimental and numerical study of forward–curved centrifugal fan", Elsevier, page no: 421-437, ISSN 0894-1777, December 2001.

[3] Sheam-Chyun Lin, Ming-Lun Tsai, done investigation on "An integrated performance analysis for a backward-inclined centrifugal fan", Elsevier, page no: 24-38,ISSN 0045-7930, 2012.

[4] Beena D. Baloni ⇑, Yogesh Pathak, S.A. Channiwala, "Centrifugal blower volute optimization based on Taguchi method",Elsevier, page no: 72-78,ISSN 112-2015, 2015.
[5] N. Vibhakar , S. D. Masutage, S. A. Channiwala, “Three Dimensional CFD Analysis of Backward Curved Radial Tipped Blade Centrifugal Fan Designed as per Unified Methodology with Varying Number of Blades”, international journal of emerging trends in engineering and development, page no: 246-256, ISSN 2249-6149, January 2012.
[6] Yogesh R. Pathak, Beena D. Baloni, Dr.S.A.Channiwala, “Numerical Simulation of Centrifugal Blower using CFX.”IJECSCSC, page no: 242-247, ISSN 2277-9477, April 2012.
[7] Peter GASPAROVIC, Maria CARNOGURSKA, “Aerodynamic optimisation of centrifugal fan casing using CFD.”, journal of applied science in the thermodynamics and fluid mechanics, page 1-6, volume 2, ISSN 1802-9388, 2008.
[8] K. Vasudeva Karanth and N. Yagnesh Sharma "CFD Analysis of a Centrifugal Fan for Performance Enhancement using Converging Boundary Layer Suction Slots", world acadamy of science, engineering and technology, page no: 381-387, ISSN 60-2009, 2009.
[9] Wan-Ho Jeon “A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan”, science direct, page no: 221-230, ISSN 265-2003, 2002.
[10] Denial wolfrom, Thomas H. Carolus “Experiment and numerical investigation of unstudy flow field and tone generation in an isolated centrifugal fan impeller” , Elsevier, page no: 1-18,ISSN 4380-4397, 2010.
[11] D.V. Bhope, P. M. Padol “experimental and theoretical analysis of stresses noise and flow in centrifugal fan impeller” , Elsevier, page no: 1257-1271 ,ISSN 39-2004, 2004.
[12] John D. Anderson, Computational Fluid Dynamics, Jr. McGraw – Hill Education (India) Private Limited.