International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 9, Issue 2, pp: (11-17), Month: October 2021 - March 2022, Available at: www.researchpublish.com

Analysis and Simulation of Axial Fan Using CFD

L. H. Kenmogne Cheteu[1], H-S. Dou[2]

Faculty of Mechanical Engineering and Automation

Zhejiang Sci-Tech University, Hangzhou, Zhejiang, 310018, China

Abstract: **The aim of this study is to analyze a nine-bladed axial fan, standard specifications are used to generate a framework of axial fan efficiency. Using currently available CFD programs, the research is carried out in an efficient manner. At the inlet, results were obtained using typical boundary conditions, with fluid conditions at the outlet. The results showed a higher pressure of 381.62 Pa and a maximum velocity of 37.5 m/s at the outlet for these conditions. These results can also be used to compare the performance of a nine-blade axial fan with a nine- and twelve-blade axial fan.**

Keywords: **Axial Fan, Computational fluid dynamics (CFD), Pressure, Velocity streamline.**

I. INTRODUCTION

The axial flow fan is widely applied in a wide range of applications in engineering. Small cooling fans for electronics to large wind tunnel fans, this type of fan is utilized in a wide range of applications. In the air conditioning and industrial process industries, axial flow fans are widely used. As a result of its adaptability, it has been included into large-scale systems, ranging from industrial dryers to automobile engine cooling and in-cabin air-recirculation systems.[1][2]

Researchers have been studying the performance of axial flow fans for fluid movement and heat transmission for many years. The performance of axial fans and their flow properties have been quantified using numerical simulations. Linear fans, or axial fans, blow air in a single direction along the axis, thus its name.[3][4]

CFD has gained in popularity over the years due to its expressive computing capabilities and substantial research in the simulation field. Complex three dimensional geometries of equipment may be simulated now with just minimal simplifications according to the use of computational fluid dynamics (CFD). These models, if they are designed effectively, consider the complex flow processes that exist inside of gear. To analyze the flow behavior and features of axial fans, CFD models have been used. A wide range of operating conditions may be expected using the models.[5]

This is difficult using other methods. Axial fan flow was modeled using CFD in this research. The goal was to figure out how to improve efficiency. While axial flow fans are incapable of producing high pressures, they are ideally suited to handle huge quantities of air at low pressures.[6]

In general, they are inexpensive and efficient, and their blades can be shaped like an aerodynamic profile. Axial fans operate on the simple principle of airflow deflection. The flow has two main components: axial and tangential velocity. If you want to move the air to/from a specific location, you need the axial velocity. The tangential velocity is either an energy loss in the fans or converted to static pressure as in the case of vane axial fans.

CFD software is used to model and analyze the collected data. The IGES files are imported into the software available for the design of the axial fan blades. These outlines (such as pressure contours and velocity contours) are computed for a nine-bladed axial fan under steady-state conditions in the present works study.[7]

II. MATERIALS AND METHODS

A. *Analysis of nine-bladed axial fan Stator*

The first step is to identify an axial flow fan that can be replicated in the form of a 3D Computer Aid Design (CAD) model and Solidworks software Fig. 1 These 3D models are then imported into Computational Fluid Dynamic software, remodeled into distinct sections, and refined to generate a finite volume mesh.

This is an essential step, where the details of the geometric structure must be accurately described. In addition, the flow domain is created Fig. 2, and the final mesh of all elements must be accurate. Any errors in this design and the flow domains must be corrected before proceeding.

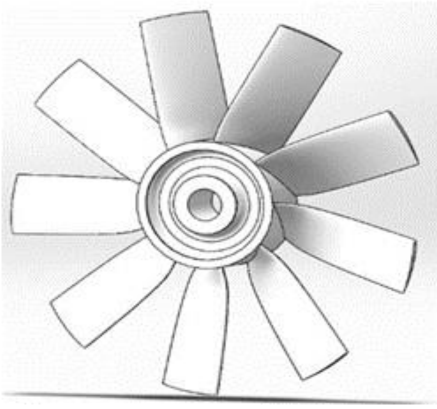

Fig. 1: Nine bladed axial fan blades mesh

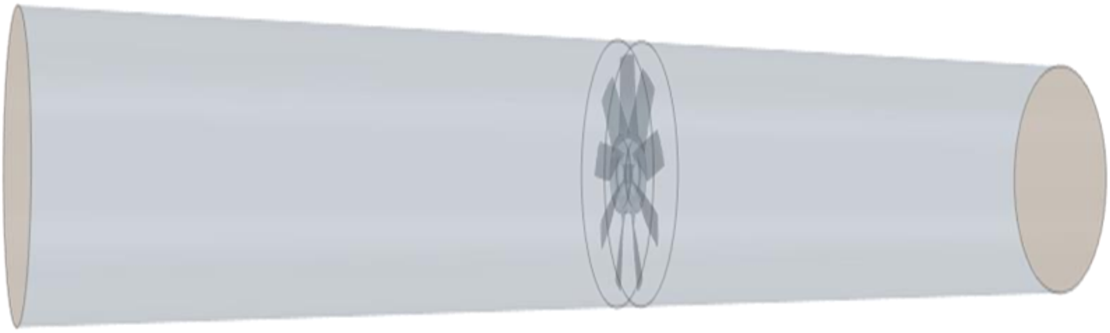

Fig. 2: Computational domain of fan

A Computer Aid Design model with a tilt ± 0.01 m is loaded into the Ansys Fluent software. The preferences specified by the user are taken into consideration during the meshing process. All concerns have been taken into consideration, as can be seen in Fig.3 from this we can assess whether the meshing procedure was acceptable or not.

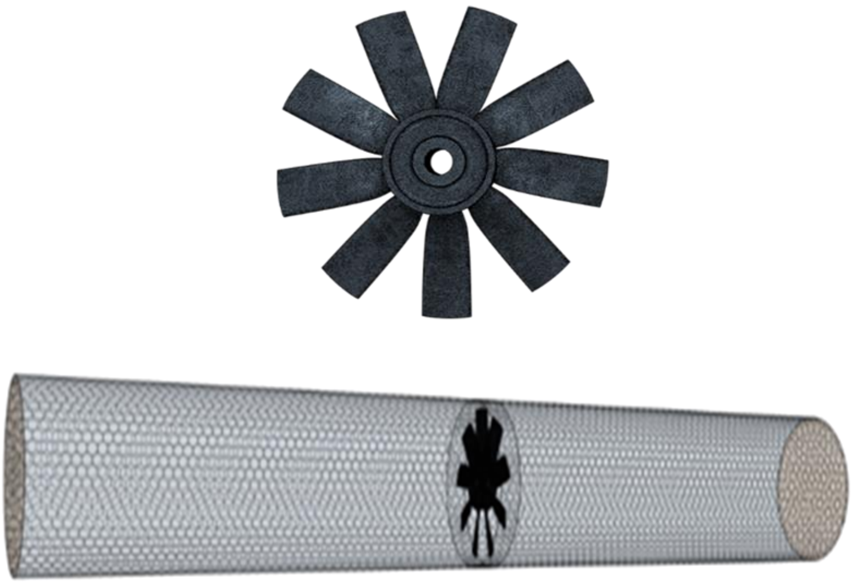

Fig. 3: Meshing Process

Before creating the mesh, it is critical to name certain components of the model, such as the inlet, outlet, boundary layer, and blade. This enables the use of fundamental laws and boundary conditions in Fluent. The mesh design is then loaded into Fluent Setup to assess mesh quality, and the mesh quality may be verified using the validation option.

The next step is to choose a model, in this case, a constant analysis should be performed, and the result is pressure-based since we require the pressure contour. The absolute velocity is specified in the following option. At ordinary atmospheric pressure, the flow is considered to be perfect dry air. The airflow is modeled using the (k-ε) computational fluid dynamic analysis with a typical wall function. Pressure is set to atmospheric and the temperature is set to 298 K as a default value. There is a pressure-based implicit in three dimensions with steady state conditions. First-order upwind is used as the solution technique. As a result, the default value for the convergence criterion is 1e-4.

Selection of the boundary conditions is made for the specified selections. Then, the solution is initialized, the computation is executed, and the number of iterations is determined. As the modeling and result expected are a bit complex, the iterations are set in 1000s and so on. Based on the current design on which the analysis is performed, it may take some time to finish the computation.

B. Validation of Numerical Analysis

H. Kumawat, et al. have compared the pressure distribution derived from the numerical analysis with their own results. H. Kumawat, et al. used seven blades, while the present analysis uses nine blades.

C. Results and Discussion

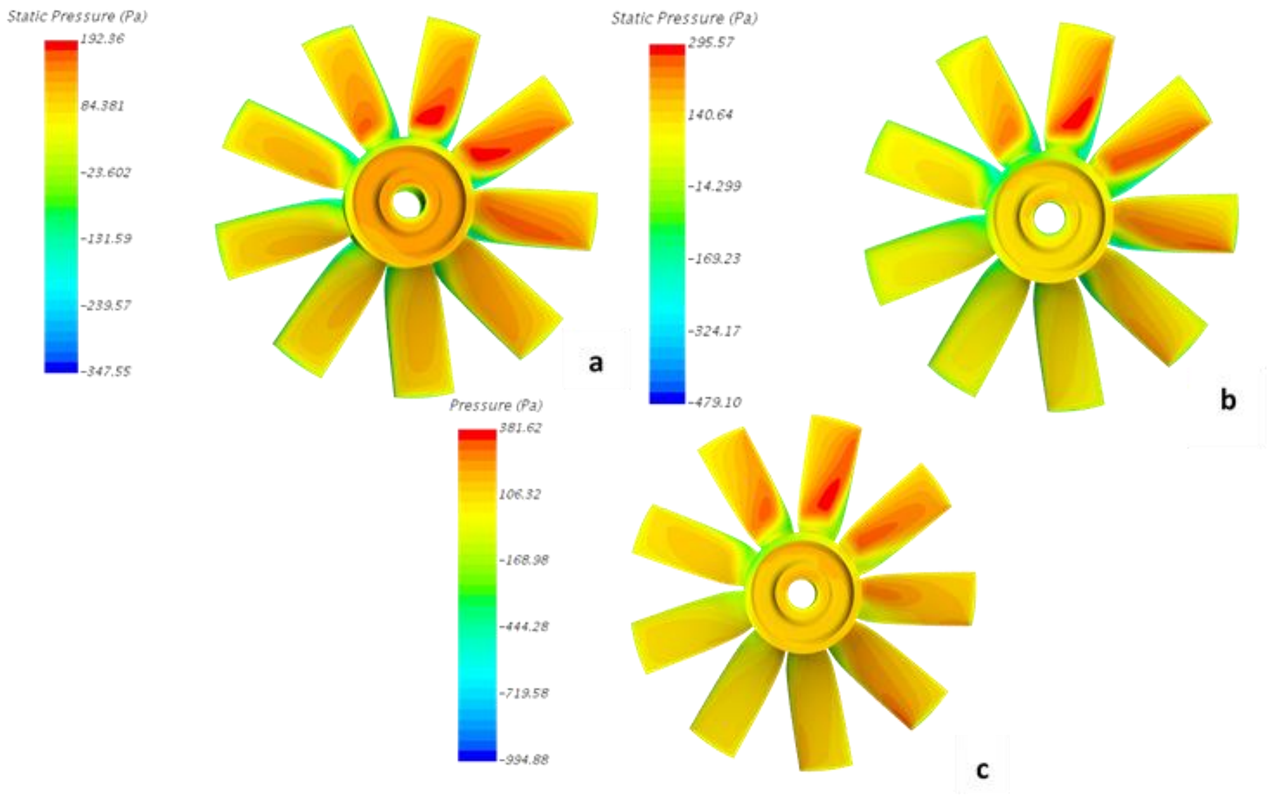

Fig. 4: Pressure contour (a) 1000 r.p.m, (b) 1800 r.p.m, (c) 2500 r.p.m

Fig. 4 shows the pressure contour of the blades while they are remained stationary. Why is pressure displayed? To better understand the pressure acting on fan blades and the hub under turbulence, the pressure is shown. In any number of blades, this may be seen. In contrast, the optimal pressure will increase as the flow entryway is reduced, resulting in greater pressures. Therefore more blades there are, the more difficult it is for air to flow between the blades of the fan. This was also 0.01m for FD1, which is based on the enclosure's diameter. Its highest pressure is 361.62 Pa and its minimum pressure is - 994.88 Pa.

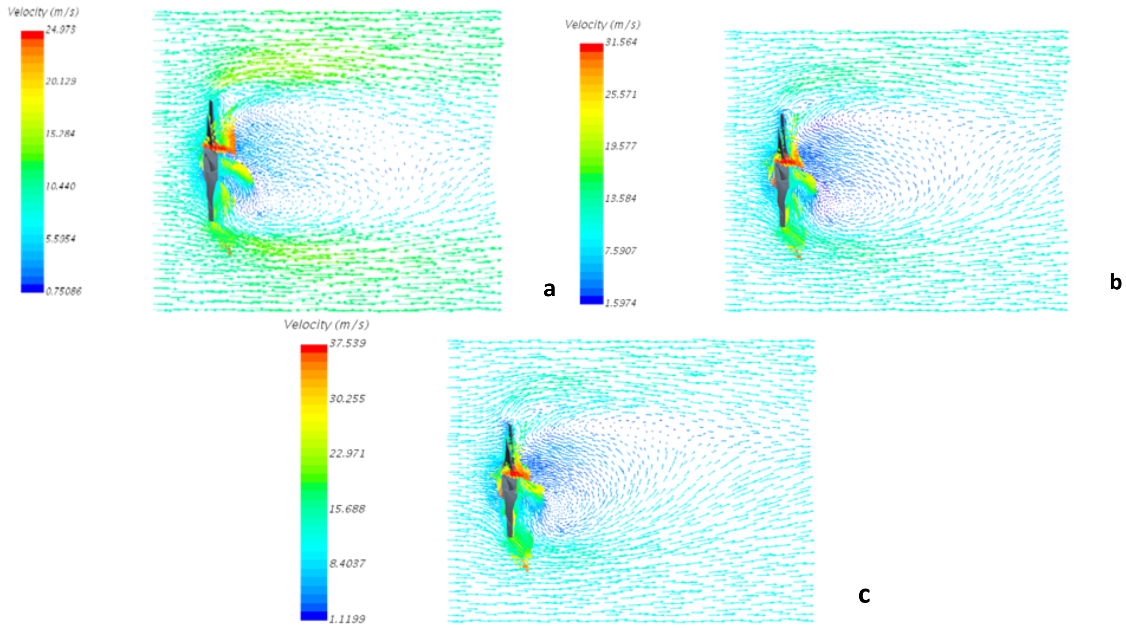

Fig. 5: Velocity vector of fan (a) 1000 rpm, (b) 1800 rpm, (c) 2500 rpm

The isometric view of the velocity vector is shown in fig. 5. Each figure has an input and an outlet. In this way it is possible to see how the airflows after is becoming uniform coming into contact with the axial fan. As observed movement is readily evident after coming in contact with the blades.

During the analysis of an enclosure, the air comes into contact with the enclosure. The highest velocity is 37.539 m/s, while the lowest velocity is 0.750 m/s. After the air has reached the blades, the highest velocity is computed, when at the blade-hub junction, the blade-hub velocity is the lowest.

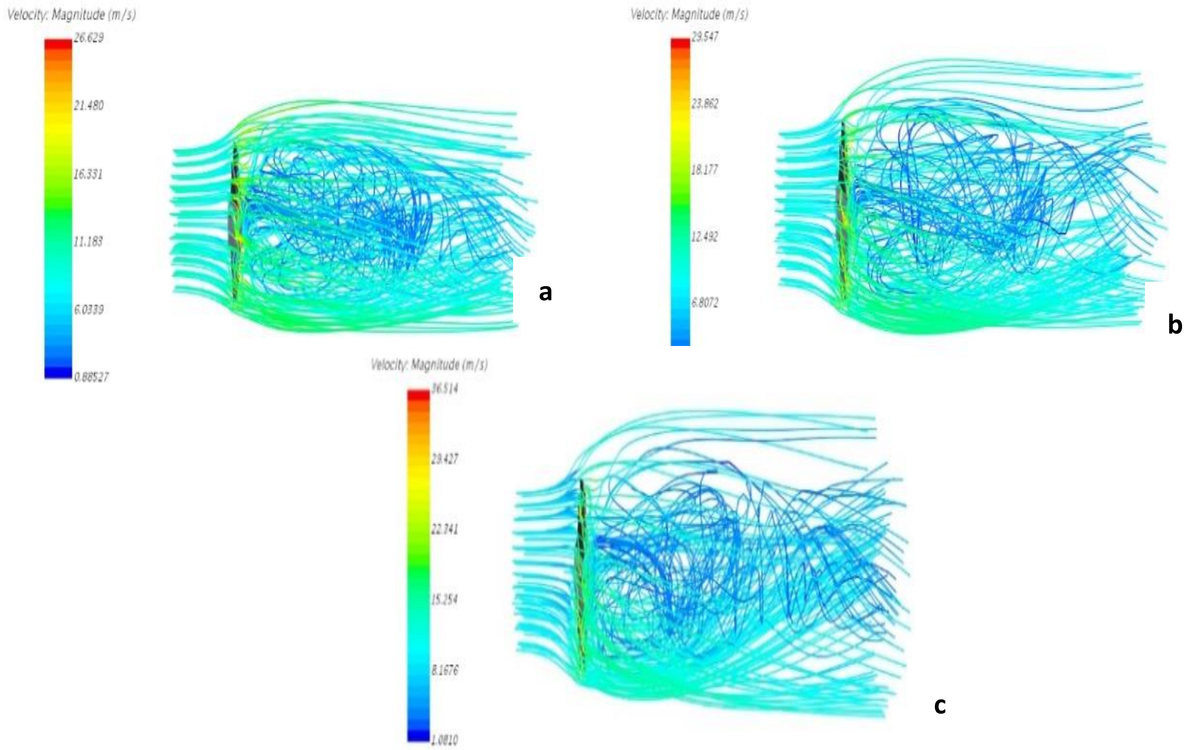

Fig. 6: Velocity streamlines (a) 1000 rpm (b) 1800 rpm (c) 2500 rpm

Fig. 6 shows the velocity streamline of the present study and the speed streamline of the present analysis. As it's obvious that the flow is whirling, it's easy to see why. The velocity at a stationary frame is the subject of the streamlined analysis. A similar result may be reached by using the velocity vector. This streamline has a highest velocity is 36.504 m/s, and a lowest velocity is 0.885 m/s. Interesting is the air's movement when it comes into touch with the blade hub. The fan should be set at a distance of 0.5m from the intake and 1m from the output, according to the designer. The originally designed fan is having nine blades. For a designed fan, outcomes had been compiled for air flowing at a rate of 22 m/s and having the outlet pressure as atmospheric. Fig. 6 shows the turbulence kinetic energy contour plot of the designed fan.

Fig. 7 : Turbulence K.E. contour

Fig. 7 shows the pressure contour of the designed axial fan. By observing the pressure contour, pressure ranges from a poor to an effective scale; hence, developing a pressure sector at the outlet.

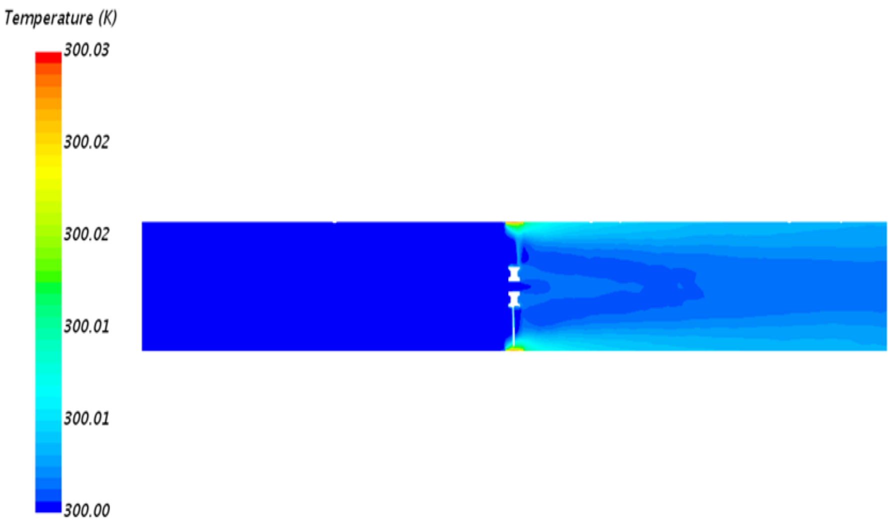

Fig. 8: Temperature contour

Fig. 8 indicates the temperature contour of the originally designed axial fan. Variation in temperature occurs because of room temperature of air and frictional heating. Temperature variation is no longer uniform on the blade surface as considered in the figure. Color (showing temporary contour) is significantly changing.

Sudden change in temperature on the blade surface will lead to the formation of thermal cracks, which can harm the blade. Moreover, the airfoil design of the blade receives distorted due to the high-temperature melting. Blade's life is incredibly decreased. The temperature of the output air is also improved efficiency goes down.

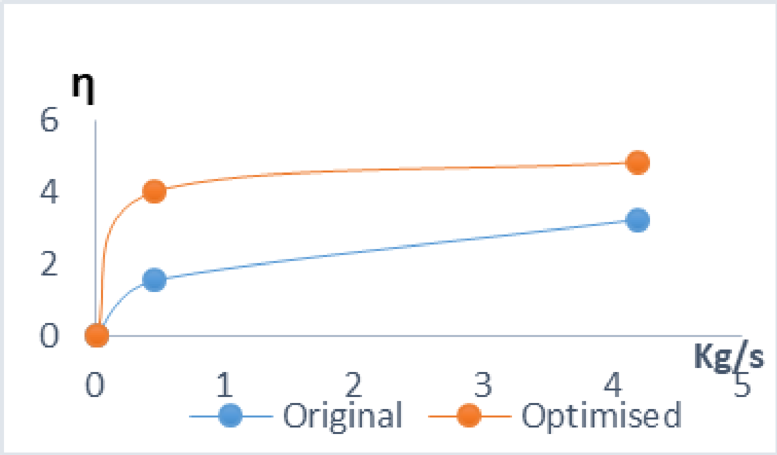

Fig. 7: Graph of RPM v/s Mass Flow

From the graph, it is found that Mass flow rate of fluid is directly proportional to the speed of the axial fan. As the axial fan, speed increases the stator outlet mass flow rate increases.

The Computational Fluid Dynamic results are presented as pressure contours, vectors (stator velocity), and velocity streamlines in the post-processing of the numerical Computational Fluid Dynamic result, respectively.

As shown in Fig. 8, the velocities are plotted against the number of iterations. We base the number of iterations on both convenience and intensity of results. The purpose of the research is unknown, but numerical simulation can establish the range of acceptable results. Iterations were limited to 4500 due to computer systems and time restrictions.

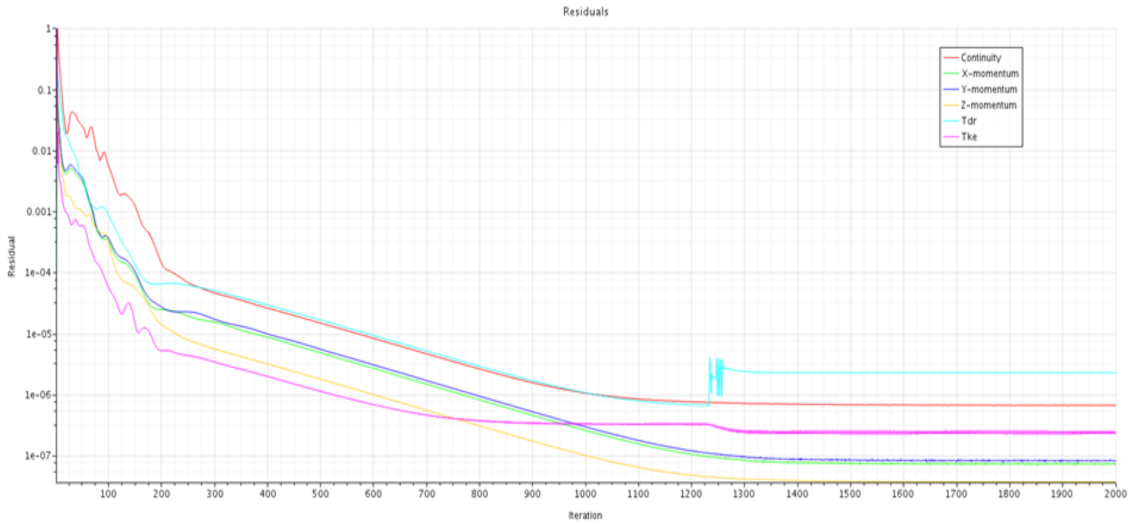

Fig. 8: Residuals v/s number of iterations rpm

D. Acknowledgement

The authors would like to express gratitude to the Government of the People's Republic of China for providing the scholarship for the leading author. This research is supported by the research fund by Zhejiang Sci-Tech University (21022094-Y).

III. CONCLUSION

The results from the numerical simulations furnished an insightful perception of the behavior of fluid flow around an axial fan with distinct number of fan blade CFD analysis was performed for each initially designed and optimized designed axial fan.

The numerical CFD results for optimized design were then in contrast with the initially designed axial fans. The key and necessary results of this find out about are as follows:

- The CFD modeling proven in this study proved to be very useful in initiating a in addition and more complete numerical study of the axial fan.
- CFD results have been introduced in the form of velocity streamlines, which supplied actual flow characteristics of air around the fan for a distinct number of fan blades.
- The distinct parameters like temperature, pressure, fan noise, and turbulence, have been also considered while performing CFD analysis. The study revealed that a fan with an most effective number of fan blades carried out properly as in contrast to the fan with much less number of fan blades. In general, as a compromise between efficiency and cost, five to twelve blades are good realistic solutions.
- The pressure contour has a maximum value of for 1000 r.p.m is159.50 Pa, for 1800 r.p.m is 295.57 Pa, for 2500 r.p.m is 381.62 Pa and the minimum value for 1000 r.p.m is -347.55 Pa, for 1800 r.p.m is -479.10 Pa, for 2500 r.p.m is -994.88 Pa.
- Velocity magnitude and velocity streamline help understand the movement of the fluid flow in an enclosed cylinder.
- The velocity in a stationary body for each vector and streamline has a maximum value of 24.97 m/s for 1000 r.p.m, 31.56 m/s for 1800 r.p.m, 37.53 m/s for 2500 r.p.m.
- The outcomes acquired are in contrast with a distinct number of blades with minor adjustments in the result.

REFERENCES

[1] S. Jain, and Y. Deshpande, "CFD Modelling of a Radiator Axial Fan for Air Flow Distribution," World Academy of Science, Engineering and Technology, vol. 6, no. 11, pp. 2445-2450, 2012.

[2] M. N. Vandana, and P. S. Sanjay, Axial Flow Fan using Ansys," Engineering and Technology, E- ISSN 0976, Vol. 2, no. 2, pp. 261-270, 2011.

[3] J. S. Patel, and S. M. Patel, "Parameter Affecting the Performance of Axial Fan Performance," International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 3, ISSN: 2278-0181, vol. 1, no. 3, 2012.

[4] K. Bamberger, and T. Carolus, "Optimization of Axial Fans with Highly Swept Blades with Respect to Losses and Noise Reduction," University of Siegen, Paul-Bonatz-Strasse 9-11, 57223 Siegen, Germany, Noise Control Engineering Journal, vol. 60, no. 6, pp. 716-725, 2012.

[5] J. Park, "A Sound Method for Fan Modelling", Fluent News, US. 2005.

[6] Moore Fans LLC, 800 S. A report on fan design by Moore, Missouri Ave., Marceline, MO, USA, TMC-661-Rev-B-12/01, pp. 7, 1940.

[7] H. Kumawat, "Modelling and Simulation of Axial Fan Using CFD," World Academy of Science, Engineering and Technology, vol. 8, no: 11, pp. 1892-1896, 2014.