ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

CFD Analysis of a Radiator Axial Fan

Mr Masanpalli Raju
M.Tech Student
Department of Mechanical Engineering ,
Mallareddy College of Engineering.

Mr Veera Naagulu, M,Tech
Assistant Professor,
Department of Mechanical Engineering ,
Mallareddy College of Engineering.

Mr C Shashikanth, M.Tech
Assistant Professor,
Department of Mechanical Engineering ,
Mallareddy College of Engineering.

Abstract

The fluid mechanics principle is used extensively in designing axial flow fans and their associated equipment. This paper presents a computational fluid dynamics (CFD) modeling of air flow distribution from a radiator axial flow fan used in an acid pump truck Tier4 (APT T4) Repower. This axial flow fan augments the transfer of heat from the engine mounted on the APT T4. CFD analysis was performed for an area weighted average static pressure difference at the inlet and outlet of the fan.

Pressure contours, velocity vectors, and path lines were plotted for detailing the flow characteristics for different orientations of the fan blade. The results were then compared and verified against known theoretical observations and actual experimental data. This study shows that a CFD simulation can be very useful for predicting and understanding the flow distribution from a radiator fan for further research work.

Keywords: ***Computational fluid dynamics (CFD), acid pump truck (APT) Tier4 Repower, axial flow fan, area weighted average static pressure difference, and contour plots***

1. Introduction

THE axial flow fan is extensively used in many engineering applications. Its adaptability has resulted in implementation into large scale systems, from industrial dryers and air conditioning units to automotive engine cooling and in-cabin air recirculation systems. The benefit of using axial flow fans for the purpose of augmenting heat transfer is particularly evident in the automobile industry because of the need for relatively compact designs. The extended use of axial flow fans for fluid movement and heat transfer has resulted in detailed research into the performance attributes of many designs [1], [2].

Numerical investigations have been performed to quantify the performance of axial fans and their flow characteristics [3], [4]. However, the more-practical example of cooling a heated engine or heated plate using an axial flow fan has received more attention in regards to understanding flow characteristics and heat transfer [5]-[7]. Moreover, an additional practice for monitoring axial fan performance is by using the experimental technique discussed in this paper. With the expressive computer capability and extensive development in the simulation field, CFD have drawn attention in recent years. With the help of CFD, the complex 3-D geometries of equipment can now be modeled with only minor simplifications. CFD models, if created correctly, can **2.** account for the complex flows in equipment. CFD models for axial fans have been used to evaluate the flow behavior and characteristics. The models provide sufficiently accurate predictions over a range of operating conditions, which are not possible using other methods. Without an understanding of the characteristics of air flow passing through a fan, problems related to engine cooling systems can never be fully resolved. In this paper, CFD were used to model the flow passing through a radiator fan, which was then compared with actual experimental data.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

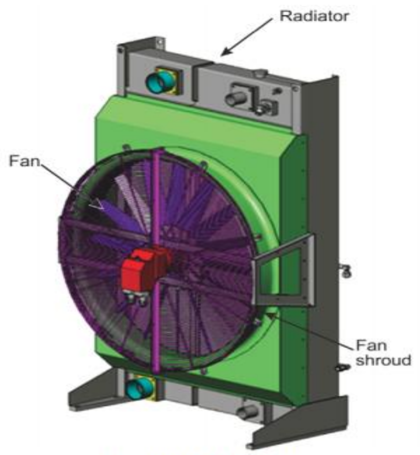
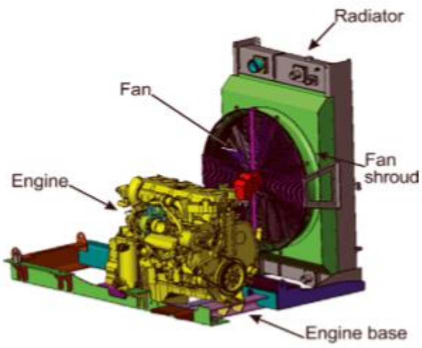


Fig. 1 Radiator and fan assembly

An APT T4 repower radiator fan and fan shroud (Fig. 1) play a crucial role in complicated engine cooling systems, such as the one shown in Fig. 2. A radiator (Figs. 1 and 2) is a type of heat exchanger designed to transfer thermal energy from the coolant to the surrounding air by means of a mechanism known as natural or forced convection. The latter case concerns the use of a radiator fan to pull the air through the radiator core.



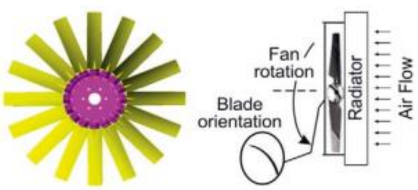


Fig. 3 Right oriented blade

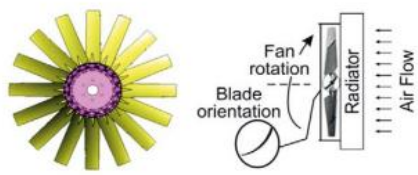


Fig. 4 Left oriented blade

The fan provides air flow through the radiator. The orientation of the blade also plays an important role in understanding the flow of air across the radiator and fan. Figs. 3 and 4 show a fan and its associated orientation configuration. For a right oriented blade, the direction of fan rotation is clockwise; for a left oriented blade, the direction of fan rotation is counterclockwise.

DESIGN AND CFD ANALYSIS OF RADIATOR FAN ASSEMBLY

The first step is to identify a typical radiator axial flow fan that can be reproduced as a 3-D CAD Solidworks® software engineering drawing package (Fig. 2). The 3-D models are then imported into the CFD software, remodeled into different sections, and refined to generate a finite volume meshing (Fig. 3). This is a crucial step, where details of the geometrical shape need to be defined precisely. The flow domain is also created, and the final meshing of all components needs to be accurate. The total element count will be around 1.6 million, with an inflation layer on the blades.

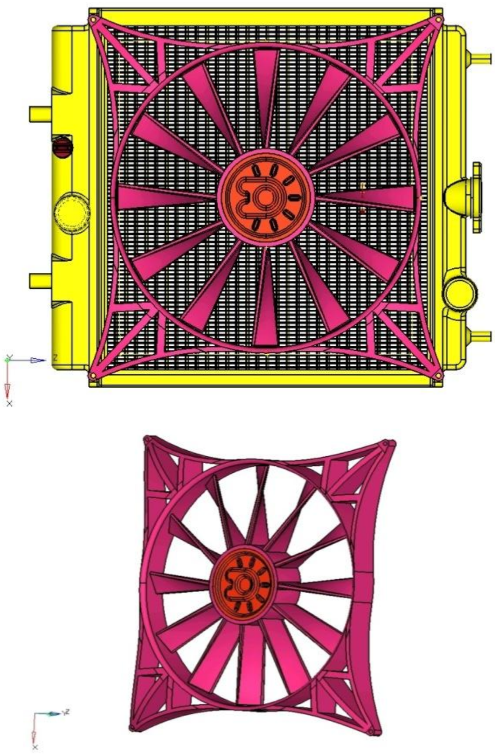


Fig 3: Computational domain of a fan

IJMETMR
ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

The second step is to import the files into the CFD code preprocessor, which will solve the flow equations. Here, the flow fields boundary conditions are set. These include inlet air mass flow, outlet pressure, fluid properties, and flow domain characterization, such as moving internal zone and stationary solid walls. The next step is to set the simulation process as a 3-D steady and turbulent problem.

Analysis results for Case-1, 2700 RPM

Fig. 7.1a, illustrates the velocity magnitude on the rotor, whichconfirms that velocity increased moving from the hub to the tip on the rotor and thus validated the theoretical concept of V= r.ω. This also affirms that the rotor was rotating at the center point of the fan axis.maximum tip velocity is 40.53m/s.

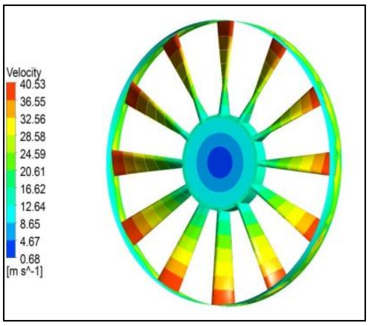


Fig. 4.1a Velocity magnitude on rotor

Figs. 4.1b show the velocity vector distribution, at a plane normal to the y-axis(outlet). A high flow region formed around the outer diameter of the flow domain and a low reverse flow region formed in the center

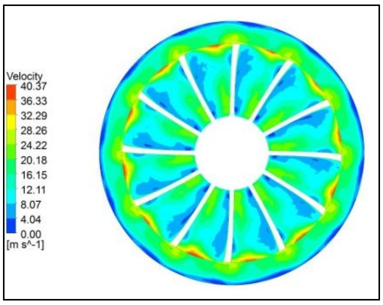


Fig. 4.1b Velocity magnitude at outlet

behind the fan hub. Between the high and low reverse flow regions, there existed strong circulation vortices. Strong circulation regions were also observed behind the fan blades. This helps in understanding the flow behavior around the rotor.

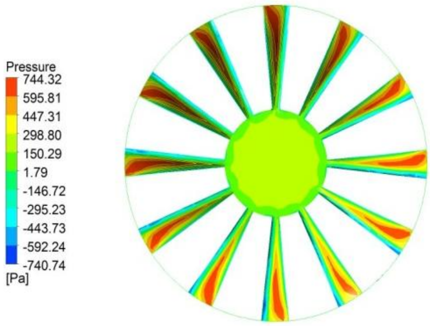


Fig. 4.1c Static pressure on rotor

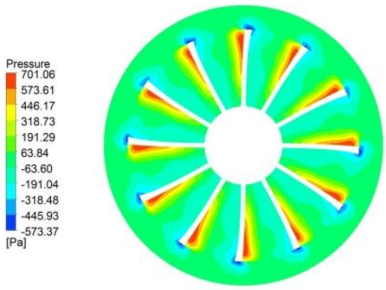


Fig.4.1d Static pressure at outlet

Figs. 4.1cand 4.1d, show the pressure contours for static pressure at outlet and on rotor. By observing the pressure contour at the rotor,pressure varies from -740.74 pa to 744.32Pa.

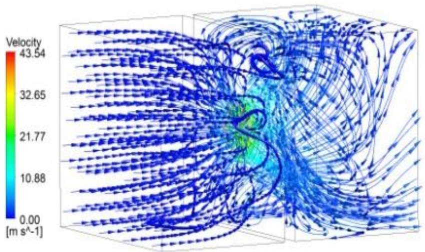


Fig. 4.1e Stream line

Fig.4.1e show the velocity distribution of streamlines from inlet to outlet of the domain. Again, a high flow region formed around the outer diameter of the flow domain (i.e., at the tip side of the blade). Also, a low reverse flow region formed in the center behind the fan hub. There existed strong circulation vortices in between the high and low reverse flow regions.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Analysis results for Case-2, 3000 RPM

Fig. 4.2a, illustrates the velocity magnitude on the rotor, whichconfirms that velocity increased moving from the hub to the tip on the rotor and thus validated the theoretical concept of V= r.ω. This also affirms that the rotor was rotating at the center point of the fan axis.maximum tip velocity is 45.04m/s.

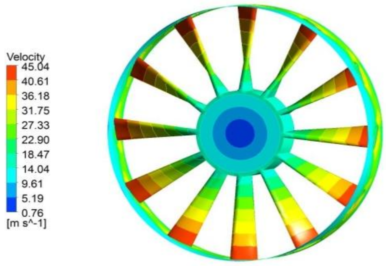


Fig. 4.2a Velocity magnitude on rotor

Figs. 4.2b show the velocity vector distribution, at a plane normal to the y-axis(outlet). A high flow region formed around the outer diameter of the flow domain and a low reverse flow region formed in the center

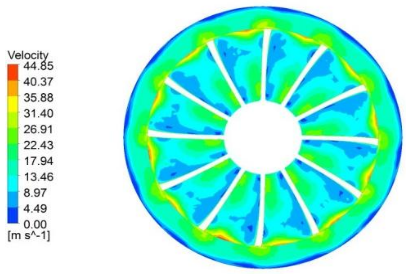


Fig. 4.2c Static pressure on rotor

behind the fan hub. Between the high and low reverse flow regions, there existed strong circulation vortices. Strong circulation regions were also observed behind the fan blades. This helps in understanding the flow behavior around the rotor.

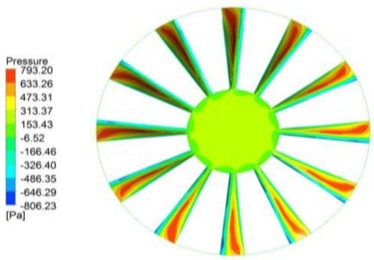


Fig. 4.2c Static pressure on rotor

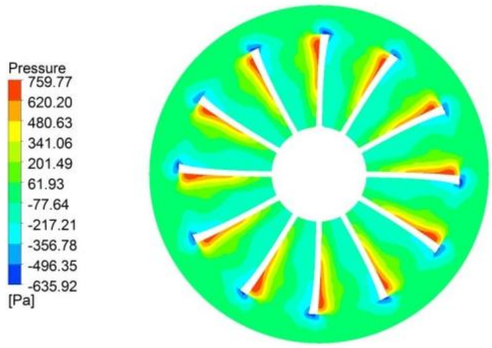


Fig. 4.2d Static pressure at outlet

Figs. 4.2c and 4.2d, show the pressure contours for static pressure at outlet and on rotor. By observing the pressure contour at the rotor,pressure varies from -806.23 pa to 793.2Pa.

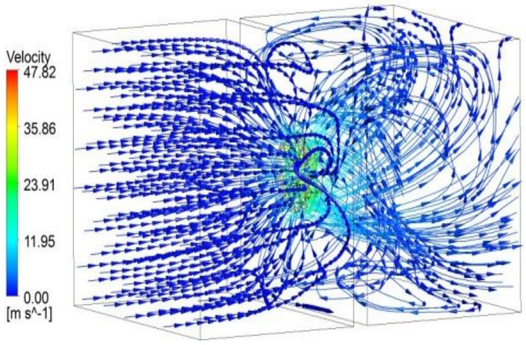


Fig. 4.2e Stream line

Fig.4.2e show the velocity distribution of streamlines from inlet to outlet of the domain. Again, a high flow region formed around the outer diameter of the flow domain (i.e., at the tip side of the blade). Also, a low reverse flow region formed in the center behind the fan hub. There existed strong circulation vortices in between the high and low reverse flow regions.

Analysis results for Case-3, 3300 RPM

Fig. 4.3a, illustrates the velocity magnitude on the rotor, whichconfirms that velocity increased moving from the hub to the tip on the rotor and thus validated the theoretical concept of V= r.ω. This also affirms that the rotor was rotating at the center point of the fan axis.maximum tip velocity is 49.54m/s.

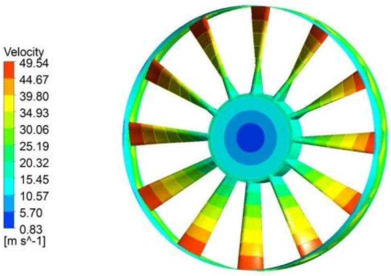


Fig. 4.3a Velocity magnitude on rotor

Figs. 4.3b show the velocity vector distribution, at a plane normal to the y-axis(outlet). A high flow region formed around the outer diameter of the flow domain and a low reverse flow region formed in the center

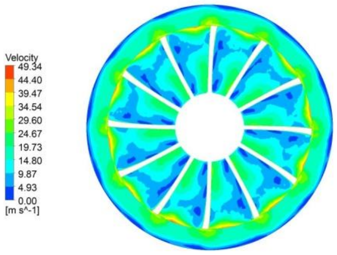


Fig. 4.3b Velocity magnitude at outlet

behind the fan hub. Between the high and low reverse flow regions, there existed strong circulation vortices. Strong circulation regions were also observed behind the fan blades. This helps in understanding the flow behavior around the rotor.

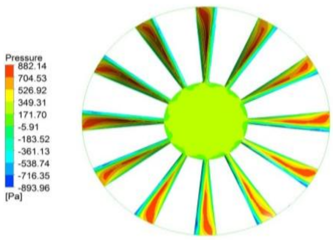


Fig. 4.3c Static pressure on rotor

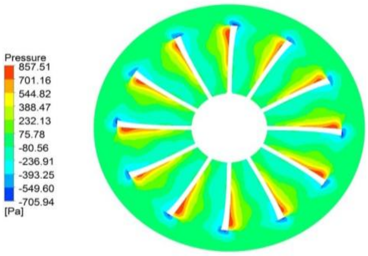


Fig. 4.3d Static pressure at outlet

Figs. 4.2c and 4.2d, show the pressure contours for static pressure at outlet and on rotor. By observing the pressure contour at the rotor,pressure varies from -893.96 pa to 882.14Pa.

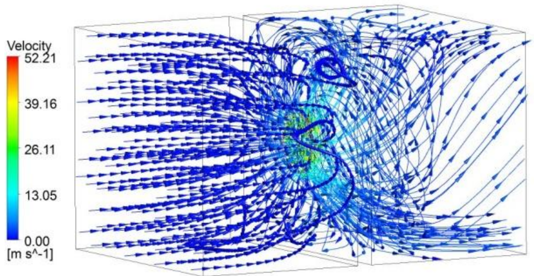


Fig. 4.3e Stream line

Fig.4.3e show the velocity distribution of streamlines from inlet to outlet of the domain. Again, a high flow region formed around the outer diameter of the flow domain (i.e., at the tip side of the blade). Also, a low reverse flow region formed in the center behind the fan hub. There existed strong circulation vortices in between the high and low reverse flow regions.

Analysis results for Case-4, 3600 RPM

Fig. 4.4a, illustrates the velocity magnitude on the rotor, whichconfirms that velocity increased moving from the hub to the tip on the rotor and thus validated the theoretical concept of V= r.ω. This also affirms that the rotor was rotating at the center point of the fan axis.maximum tip velocity is 54.04m/s.

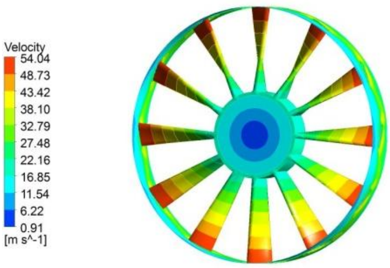


Fig. 4.4a Velocity magnitude on rotor

Figs. 4.4b show the velocity vector distribution, at a plane normal to the y-axis(outlet). A high flow region formed around the outer diameter of the flow domain and a low reverse flow region formed in the center

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

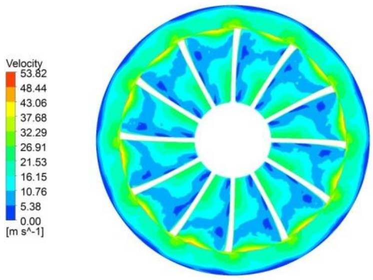


Fig. 4.4b Velocity magnitude at outlet

behind the fan hub. Between the high and low reverse flow regions, there existed strong circulation vortices. Strong circulation regions were also observed behind the fan blades. This helps in understanding the flow behavior around the rotor.

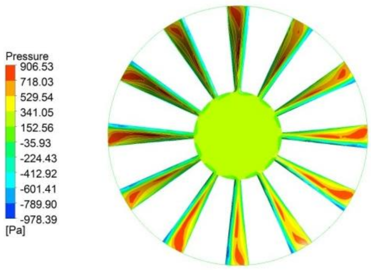


Fig. 4.4c Static pressure on rotor

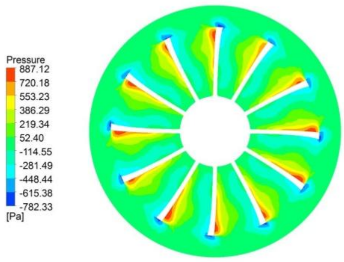


Fig. 4.4d Static pressure at outlet

Figs. 4.4c and 4.4d, show the pressure contours for static pressure at outlet of the rotor and on rotor. By observing the pressure contour at the rotor,pressure varies from -978.39 pa to 906.53Pa.

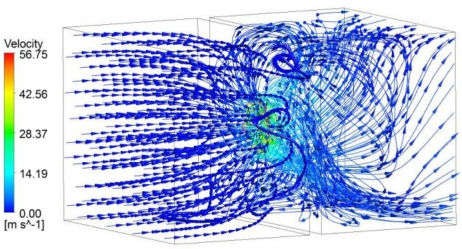


Fig. 4.4e Stream line

Fig.4.4e show the velocity distribution of streamlines from inlet to outlet of the domain. Again, a high flow region formed around the outer diameter of the flow domain (i.e., at the tip side of the blade). Also, a low reverse flow region formed in the center behind the fan hub. There existed strong circulation vortices in between the high and low reverse flow regions.

RESULTS:

On Post processing the numerical CFD results, the observations are presented as velocity vector distributions, flow lines and static pressure contour plot at mid section of the fan and also on rotor. For the velocity vector representation, a plane is taken normal to the y-axis coordinate and at the centre (0, 0, 0,). Results are compiled separately for the front and back oriented blades

4. ACKNOWLEDGEMENT

We would like to thank all the authors of different research papers referred during writing this paper. It was very knowledge gaining and helpful for the further research to be done in future.

REFERENCES

1) K. Duvenhage, J. A. Vermeulen, C. J. Meyer, and D. G. Kroger, “Flow Distortions at the Fan Inlet of Forced-Draught Air-Cooled Heat Exchanger,” in Applied Thermal Engineering, Vol. 16, Nos. 8/9, pp. 741–752, 1996.

[2] F. W. Yu and K.T. Chan, “Modelling of a Condenser-Fan Control for an Air-Cooled Centrifugal Chiller,” in Applied Energy Vol. 84, pp. 1117– 1135, 2007.

[3] E. H. Twizell and N. J. Bright, “Numerical Modelling of Fan Performance,” in Applied Mathematical Modelling, Vol. 5, 1981.

[4] J. R. Bredell, D. G. Kroger, and G. D. Thiart, “Numerical Investigation of Fan Performance in a Forced Draft Air-Cooled Steam Condenser,” in

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Applied Thermal Engineering, Vol. 26, pp. 846–852, 2005.

AUTHOR(S) PROFILE

Mr. Masanpalli Raju, M.Tech, Mallareddy College of Engineering.

Mr. Veera Naagulu, B.E, M.Tech, Assistant professor, Mechanical department, Mallareddy College of Engineering.

Mr. C. Shashikanth, B.Tech, M.Tech, Assistant professor, Mechanical department, Mallareddy College of Engineering.