RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING VOL. 2 NO. 1 (2021) 20 – 26

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Numerical Analysis of Performance on Internal Flow for Contra-Rotating Small-sized Cooling Fan

Mohammad Shahril Azman[1], **Bukhari Manshoor**[1*]

[1]Faculty of Mechanical Engineering and Manufacturing
Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2021.02.01.003
Received 15 Feb 2020; Accepted 27 March 2020; Available online 15 April 2021

Abstract: A contra-rotating fan is basically two fans combine but rotate in a different direction. Many still do not familiar with the term "contra", as the contra-fan itself is rarely used. Contra means different, which explain the two-fan blade rotating in opposite direction. The contra-rotating fan consists of the front blade that rotated in a clockwise direction and the rear blade rotated counterclockwise. A lot of energy conversion efforts has made in the past, in order to improve efficient airflow. From the previous study, it is shown that the contra-rotating fan had many advantages compared to the axial fan. counter-rotating axial-flow fans could be a promising way to achieve these requirements. This is because of the reduction of rotational speed and a better homogenization of the flow downstream of the rear rotor and produces very good aerodynamic performances. Computational Fluid Dynamic (CFD) software. The blades were designed using SolidWorks 2019 software and then been evaluate or simulate using Ansys 19.3. The contra-rotating fan setup was compared with another fan setup. The performance evaluations of contra-rotating are compared in terms of the airflow velocity parameters.

Keywords: Ansys, FLUENT, CFD, Contra-rotating

1. Introduction

In today's generation of technology, of all the challenges of electronic devices in the industry, keeping components cool is the most important since overheating causes significant reductions in the operating life and to avoid thermal failure. As we know, turbomachinery is a machine that transfers energy between a rotor and a fluid, including both turbines and compressor. The turbine will extract energy to the fluid, while the compressor will exchange energy with fluid by absorbing it. However, its entire requirement does not apply the design requirement for small-sized contra-rotating fans due to the extremely small size field compare to turbo machinery. For the small-sized contra-rotating fan, there is the limitation of space, to achieve high performance of cooling fans, it is important to clarify the complicated internal flow condition. Contra-rotating fan and known as coaxial contra-rotating basically just a two rotor (front and rear) that rotating in opposite direction were combined.

*Corresponding author: bukhari@uthm.edu.my

It supposed to produce more efficiency and minimize the effect of torque produced by each other rather than a single rotor. This application can be found mainly on aircraft propellers, it utilized the power of a single piston engine to drive two propellers that spin in opposite direction. The two propellers were arranged on the same axis, one at the front and one at the back, and its transfer the power by using a planetary gear system. Contra-rotating fans in are rarely used commercially, some ceiling fan and electrical components cooler fan utilized the mechanism, but it is not a common option. The biggest difference between conventional and contra-rotating fans is the number of rotors. Conventional fan only has one set of blades on a rotor, normally ranging from 2 to 6, rotating on an axis to produce a stream of airflow parallel to the shaft to a preferred direction to promote heat convection in order to reduce the temperature of the target area or object. Contra-rotating fans on the other hand, boast two sets of fan blades on two separate rotors spinning in opposite direction from each other.

2. Methodology

2.1 Modelling and setup

In this study, the scope is focused on simulating and comparing the airflow behaviour of both conventional and contra-rotating fan setups. Figure 1 shows the setup for the simulation while Table 1 summarise the parameter setup.

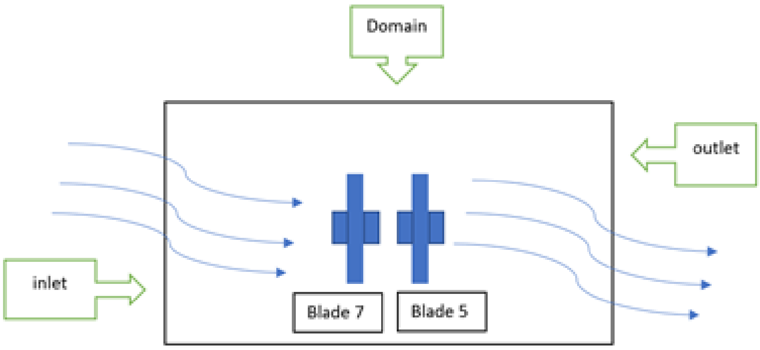


Figure 1: Setup for simulation work

Table 1: Parameter for counter rotating fan

Parameter	Front rotor	Rear rotor
Hub Radius (mm)	31.5	31.5
Tip Radius (mm)	57.2	57.2
Duct diameter (mm)	116	116
Tip clearance (mm)	0.8	0.8
Axial chord length (mm)	24	30
Rotational speed, (rpm)	3650	3150
Blade number	7	5
Solidity	1	0.95

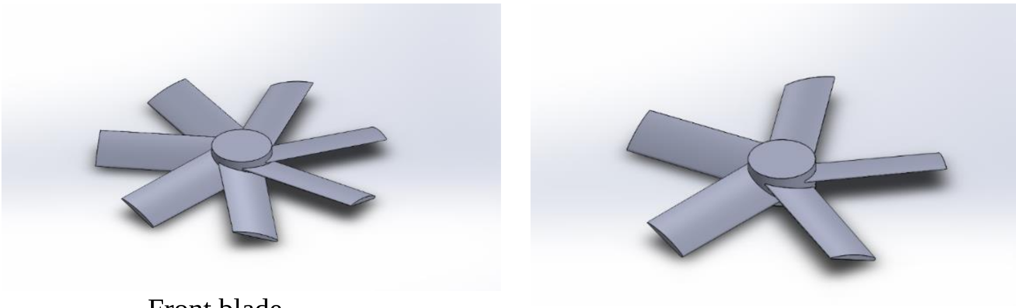
A total of 6 different fan setups are used for this study to observe and differentiate the airflow properties between different setups. The parameters of the fan setups are as shown in Table 2. The setups were contra direction, same direction and single. For the contra setup, 7 blades fan as the front rotor rotating in clockwise direction while 5 blades fan as rear rotor rotating anti-clockwise direction. For the same direction setup, 7 blades fans both rotating in clockwise direction. Lastly, for the single setup, 7 blades fan rotating in the clockwise direction. In this setup, there are variations of inlet velocity 20 m/s were used to observe the airflow pattern of the setup. A positive rotational speed indicates that the fan rotates in the clockwise direction, while a negative rotational speed means that it rotates in the anti-clockwise direction.

Table 2: Parameter for counter rotating fan

Name	No. of fans	No. of blades	Inlet velocity (m/s)	Rotational Speed (RPM)
Contra_75	2	7,5	20.0	(3650), (-3150)
Same direction_77	2	7,7	20.0	(3650), (-3150)
Single_5	1	5	20.0	(3650)

2.2 The geometry

The model was created in SolidWorks by referencing the parameters previously mentioned in Table 1 and the initial sketch in Figure 2. The front fan for contra-rotating fan setup has 7 blades while the rear fan has 5 blades. Once noticeable difference is that both fans had their leading edge facing different direction. This is to accommodate the fact that the front fan spins in a clockwise direction when viewed from the front, and the rear blade spins in anti-clockwise direction, resulting in a contra-rotating.



Front blade Rear blade

Figure 2: Geometry for blades

2.3 Meshing

Mesh is defined as the measurement of particle size often used in determining the particle-size distribution of a granular material. In fluid simulation, meshing is one of the most important steps as it determines the accuracy of the calculation outcome. The smaller and the finer the mesh size, the more accurate the result is. However, very fine mesh can result in immensely long calculation time, even for a powerful computer. It may also result in instability or even program crashes. For this study, a mesh size of 15 mm was used to maintain a balance between mesh size and calculation time. The fan blades, rotating boundary, domain, inlet, outlet, and walls are selected and added as named selection to ease the future process. To control the mesh refinement, face sizing, body sizing and inflation were added to the selected geometry.

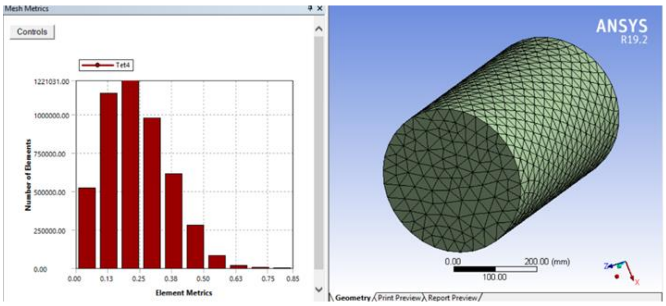


Figure 3: Skewness of the mesh created

Grid independent test was performed to evaluate the correct mesh size and number of elements that did not substantially affect the grid sensitivity of the analysis. It needs to remeshing into a smaller element size to reach the acceptable mesh size level of adaptive. The size which has been re-meshing focused only on the blade part as it is an essential element that needs to be simulated and the performance result analysed. The experimental tests on the inlet-velocity of 20 m/s were showed in Table 3

Table 3: Parameter for counter rotating fan

Element size (mm)	Number of nodes	Number of elements	Skewness
20	124101	642167	0.98
15	142164	741327	0.95
8	190735	1012913	0.96
6	816584	4595704	0.95

According to the grid independent test, the 6 mm element size was chosen due to the least skewness compared to the others. It was therefore considered to be the most reliable and appropriate element size.

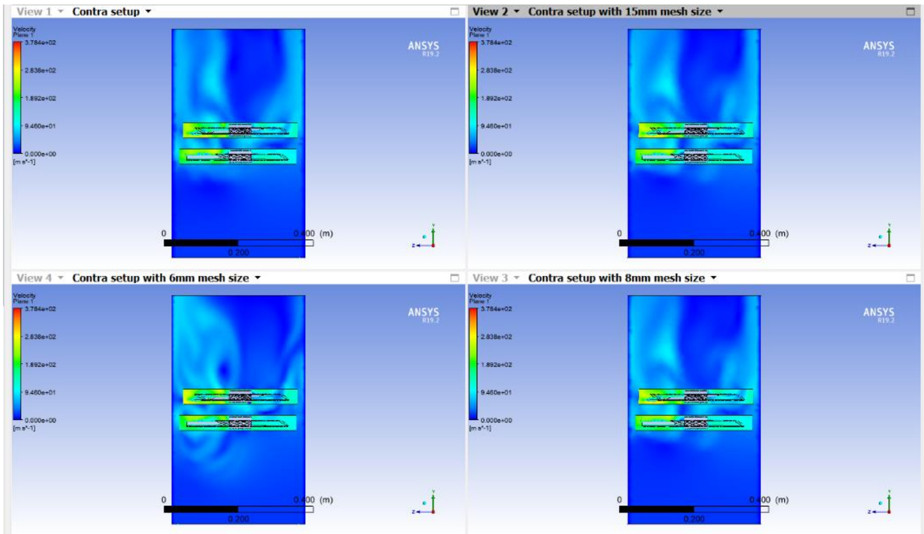
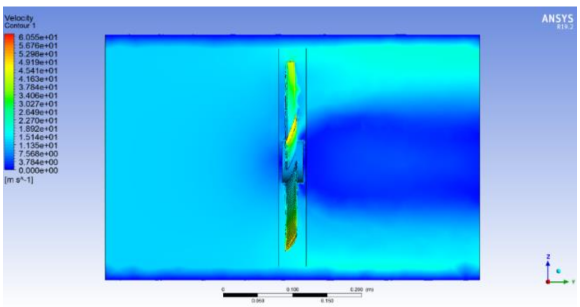


Figure 4: The comparison between contra setup with different mesh size

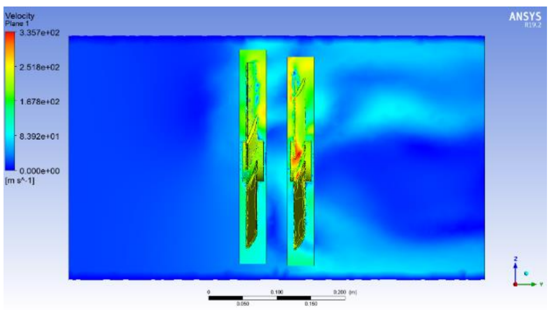
3. Results and Discussion

3.1 Velocity contour

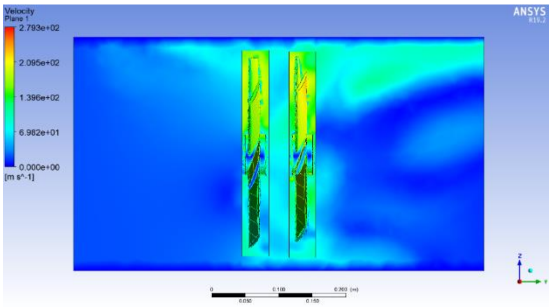
Three setups then had the inlet velocity set at 20 m/s to simulate the airflow created by spinning fans. The contour maps are taken from the last frame of the 50 times steps as before, and the results are as shown in Figure 5 below.



Max velocity: 60.55 – 56.76 m/s
Min velocity: 7.568 – 3.784 m/s
(a) Single setup



Max velocity 335.7 – 251.8 m/s
Min velocity: 167.8 – 839.2 m/s
(b) Contra setup



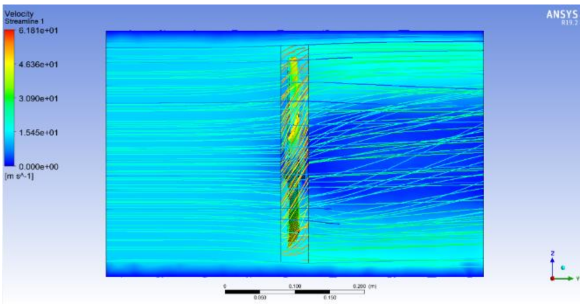
Max velocity 279.3 – 209.5 m/s
Min velocity139.6 – 69.82 m/s
(c) Same setup

Figure 5: Contour maps of the three setups for 20 m/s

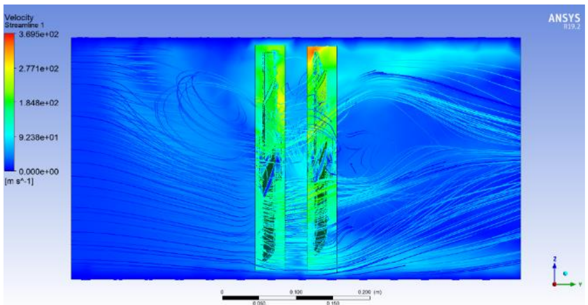
Based on the all three setups, the single setup had the uniform contour compare to the other setups, but even though the contra setup had the less uniform contour compare to the single setup, the contra setup had a larger maximum velocity ranging 335.7 to 251.8 m/s. while the maximum velocity for single setup ranging 60.55 to 56.76 m/s. Moreover, from the results between the contra and same setups, it is shown the similarity as both of the setup using two fan blades, but focusing at the maximum velocity, contra setup had the higher value compare to the same setup which ranging 279.3 to 209.5 m/s.

3.2 Streamline

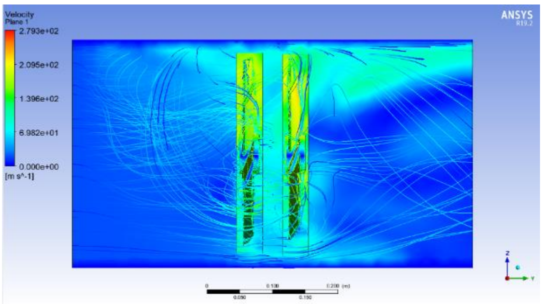
A streamline is the direction through the fluid domain that a particle with zero mass can follow. Streamlines start on a given locator at each node; in which case the streamline begins at the inlet. Another way to visualize the behaviour of airflow required for this study to be analysed is Streamline. The procedure is the same as before, using the last frame of the 50-time measurement of measures as the analysis subject.



(a) Single setup



(b) Contra setup



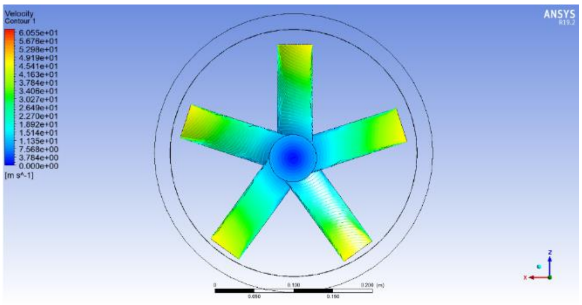
(c) Same setup

Figure 6: Streamline results of the three setups for 20 m/s

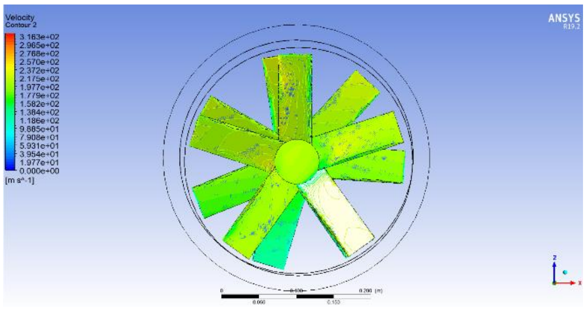
From the figures, contra setup produced a narrow, focused airflow. It is also noticed that the airflow traverse at the centre. Same and contra setups produced an airflow with similar behaviour, but the same setup's airflow less focusing at the centre. The airflow in contra setup spreads out earlier than in same setup. This is likely caused by the rear fan disrupting the flow of the air. Note that both the same and contra setup had their flow of air expanded towards the boundary, and likely to spread out even further if not limited by the certain diameter of the boundary. For the single setup, it produces a straight and wide airflow.

3.3 Velocity contour at the blades

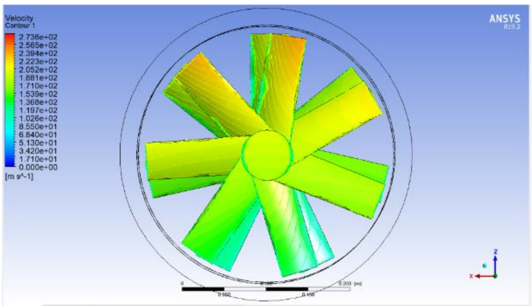
The preview of the velocity contour was set with 20 m/s inlet velocity. Generally, the velocity distribution over all wind speeds has the same gradient across the blade. The blade, absorbs energy from the inlet of the wind, faces a high velocity at the return blade, which covers the region from the inlet wind speed, faces the reduction velocity at the rotational phase. High rotation gives a high gradient of velocity around the blade.



(a) Single setup



(b) Contra setup



(c) Same setup

Figure 7: Velocity contour at the blades

From the figures, the velocity contour at the blades for all setups, First, the single setup had minimum velocity (3.784 – 7.568) m/s revolved around the hub of the 5-blades single setup, and increasing across the blades up to (34.06 – 49.19) m/s. For the contra setup, the both 7 and 5 blades had a high velocity contour at the blades which ranging (158.2 - 177.9) m/s to (237.2 – 257.0) m/s. For the same setup, the velocity spreads uniformly ranging (188.1 - 222.3) m/s.

4. Conclusion

Throughout this analysis, the approached computational fluid dynamics (CFD) is used to numerically evaluate the performance analysis of the contra-rotating fan. The simulation research was carried out using the steady procedure used to calculate the quantity. As in case of contra-rotating, the value for each time measured was dependent on the angle of rotation of the blade. It is special from the steady state approach used for just one form of angle. K-omega turbulence had been used for this solution as it can be expected well below the boundary wall. However, the simulation analysis had its own limitation because it demanded very high specification of computer. Therefore, due to this problem, the time limitation of the duration of the project would take a lot of time. The results from the numerical simulations provided a strong understanding of the internal flow around the contra-rotating fan, and. an aerodynamic airflow. This study proved the Ansys Fluent CFD modelling shown to be very helpful in processing further and more comprehensive numerical study of the contra-rotating fan.

Acknowledgement

This research was made possible by funding from research grant provided by the Ministry of Higher Education, Malaysia. The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] Fukutomi, J., Shigemitsu, T., & Yasunobu, T. (2008). Performance and internal flow of sirocco fan using contra-rotating rotors. Journal of Thermal Science, 17(1), 35–41.

[2] Luan, H., Weng, L., & Luan, Y. (2018). Numerical simulation of unsteady aerodynamic interactions of contra-rotating axial fan. PLoS ONE, 13(7), 1–21.

[3] Moroz, L., Pagur, P., Govorushchenko, Y., & Grebennik, K. (2011). Comparison of Counter – Rotating and Traditional Axial Aircraft Low-Pressure Turbines Integral and Detailed Performacnes, (August), 1–12.

[4] Nouri, H., Ravelet, F., Bakir, F., Sarraf, C., & Rey, R. (2012). Design and experimental validation of a ducted counter-rotating axial-flow fans system. Journal of Fluids Engineering, Transactions of the ASME, 134(10), 1–12. https://doi.org/10.1115/1.4007591

[5] Shigemitsu, T., Fukuda, H., & Hirosawa, K. (2017). Unsteady Flow Condition between Front and Rear Rotor of Contra-Rotating Small Sized Axial Fan. Open Journal of Fluid Dynamics, 07(03), 371–385

[6] Shigemitsu, T., Fukutomi, J., & Agawa, T. (2013). Internal flow condition of high power contra-rotating small-sized axial fan. International Journal of Fluid Machinery and Systems, 6(1), 25–32. https://doi.org/10.5293/IJFMS.2013.6.1.025

[7] Sonohata, R., Fukutomi, J., & Toru Shigemitsu, T. (2012). Study on Contra-Rotating Small-Sized Axial Flow Hydro Turbine. Open Journal of Fluid Dynamics, 02(04), 318–323. https://doi.org/10.4236/ojfd.2012.24a039

[8] Velde, Oliver & Friebe, Christian & Korfanty, Marius. (2017). Design and Optimization of Contra-Rotating Fans. https://doi.org/10.13140/RG.2.2.16004.42883

[9] Molland, A. F. (2008). Chapter 6, Marine engines and auxiliary machinery. The Maritime Engineering Reference Book, 1(1995), 344–482. https://doi.org/10.1016/b978-0-