UDK 621.313.3

DOI: http://doi.org/10.20535/2219-3804202019141784

Gurkan Ortamevzi[1], *Dr. Lecturer, Candidate of Technical Sciences*
Ibrahim Emre Gunduz[2]

ANALYSIS OF COOLING OF ELECTRIC MOTOR BY RADIAL FAN; REALIZATION OF THE EXPERIMENTAL ANALYSIS OF CFD

Uk Показан аеродинамічний аналіз вентиляторів і їх кожухів різних геометрій. Гарне обдування знижує температуру у двигуні та збільшує термін його служби. Зниження температури на 10 градусів може продовжити термін служби електричної ізоляції двигуна у два рази. Для визначення оптимальної геометрії вентилятора і його кожуха, а також оцінки її ефекту, аналіз проводиться за однакових умовах. Метою цього дослідження є визначення стану проектованого електродвигуна, який не має прототипу. Для кореляції аналізів із натурними експериментами був використаний параметр - швидкість входу повітря. Експериментальні результати можуть бути використані для проектування електродвигунів, а також для оптимізації їх аеродинамічних характеристик.

Ru Показан аэродинамический анализ вентиляторов и их кожухов различных геометрий. Хорошее обдувание снижает температуру в двигателе и увеличивает срок его службы. Снижение температуры на 10 градусов может продлить срок службы электрической изоляции двигателя в два раза. Для определения оптимальной геометрии вентилятора и его кожуха, а также оценки её эффекта, анализ проводится при одинаковых условиях. Целью этого исследования является определить состояние проектируемого электродвигателя, который не имеет прототипа. Для корреляции анализов с натурными экспериментами был использован параметр - скорость входа воздуха. Экспери-

[1] *Gazipasa Aerospace Sciences Faculty , Alanya Alaaddin Keykubat University*
[2] *İstanbul Technical University*

ментальные результаты могут быть использованы для проектирования электродвигателей, а также для оптимизации их аэродинамических характеристик.

Introduction

The purpose of this research is to simulate the cooling state of an electric motor that does not yet have a prototype. In the induction motor, energy which cannot transform to the mechanical energy will appear like heat energy. The induction motor will run as efficiently as it can be cooled. The cooling mechanism is as follows; the radial fan connected to the main shaft of the motor gives the air in the radial direction. The fan cover allows the radial air to pass over the induction motor and the heat to move to the atmosphere by convection and radiation. Convection cooling is the simplest and one of the earliest techniques used [2].

The parameters can be changed in this experiment are quite limited. For example, it cannot be made a change in the speed of rotation of the fan. Because the fan is hardly connected to the main shaft of the motor and turns at a speed equal to the speed of the shaft. In other words, it cannot be connected gear systems such as multiplier or reducer. The point can be influenced in constraints is the flow mechanism. It was ensured the optimization of the evacuation of this air flow by increasing the performance. The things it can be entered into this border; the fan itself, the fan cover, the fins on the motors. If it cannot be changed the fins, it can be made the air circulate around these fins. So it needs to be well known what direction the air is present in, speed value, flow rate and pressure. Thanks to these, it will be known the clear aspects of the improvement in the new design.

Expressed as the ratio of the inlet velocity (Vinlet) to the free flow velocity (Voutlet) with discharge condition. This ratio is the minimum value of the Vinlet/Voutlet that the vortex cannot form [3]. Because of this gap between the induction motor fan and the fan lid, it is practically not possible to create a vortex. However, it can be measured and minimized.

The actual working airflow is determined by the intersection of the fan curve and the system resistance curve. There are three options for estimating this working point: experimental measurement of the system using thermal/mechanical modeling, calculation of the operating point using airflow network methods, or calculation of the system.

The main issue for most CFD applications is the modeling of turbulence [4]. In these experiments, visual and numerical results of flow analyzes were obtained using CFD. Streamlines in particular have many perceptual benefits due to their ability to provide a snapshot of the vectors near key features of complex 3D flows at any instant in time. However, streamlines do not lend themselves well to animation [5]. So vector and contour views were used.

Material and Methods

It was used 112 frame, 1500 rpm, 5,5 Kw induction motor at these analysis and experiments.

Experimental Models:

1. Fan Models: The following three types of fans in fig. 1 – fig. 3 are designed for flow analysis.

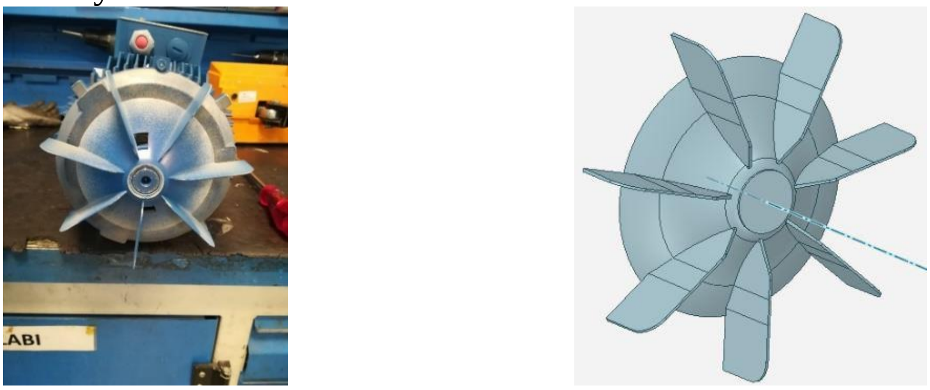

Fig. 1. Fan 1

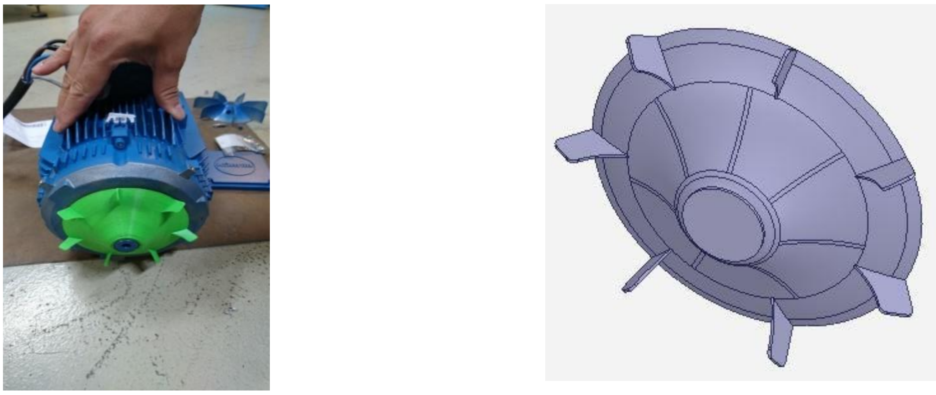

Fig. 2. Fan 2

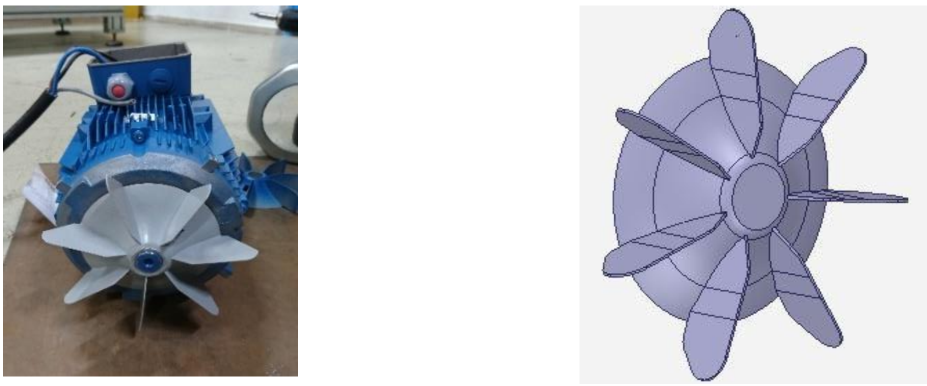

Fig. 3. Fan 3

2. Fan Cover Models: According to the results of the first experiment, despite changing fan geometry has positive effect on the flow, the negative effect created by the vortexes has also been tested in experimental models of the

fan cover designs on the elimination of the vortex formation. The following three types of fan covers in fig. 4-6 are designed for flow analysis.

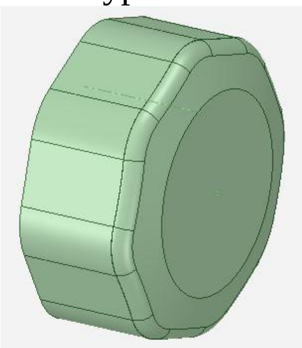
Fig. 4. Fan cover 1

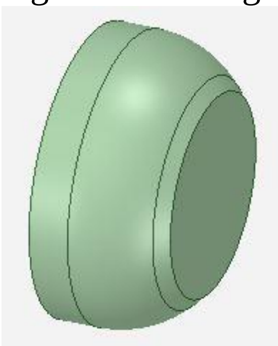
Fig. 5. Fan cover 2

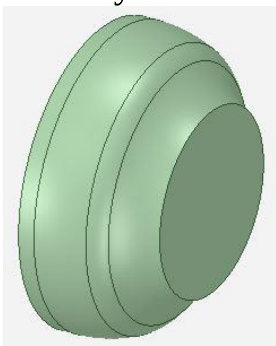
Fig. 6. Fan cover 3

Results of Calculation of CFD

1. Fan Models

Fan 1

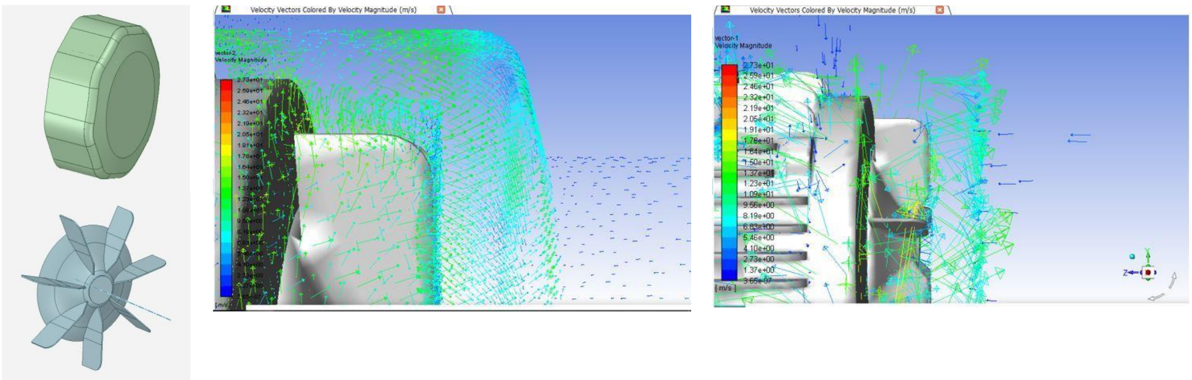

Fig. 7. Vectorial View of Result of CFD for Velocity for fan 1/fan cover 1 About Inside Fan Cover

As a result of the above analysis, it appears that some of the air does not go upright from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door.

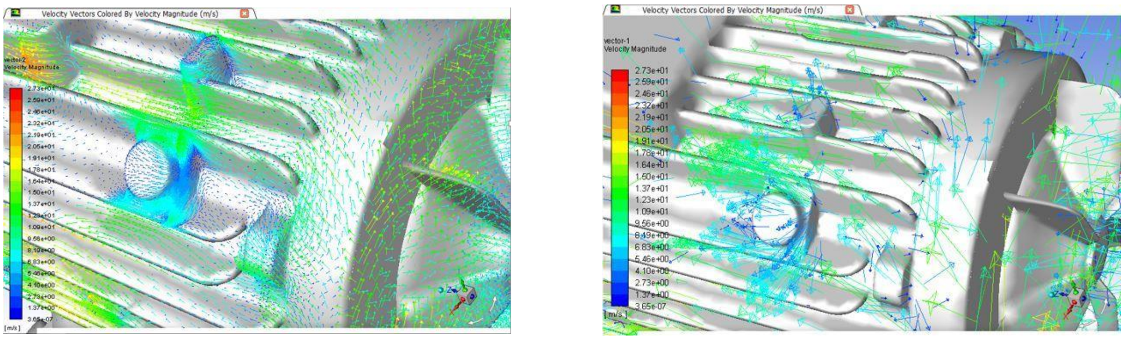

Fig. 8. Vectorial View of Result of CFD for Velocity for fan 1/fan cover 1 Over Induction Motor Body

As a result of the above analysis, it is seen that the air has lost its velocity and direction by hitting the protrusions on the induction motor.

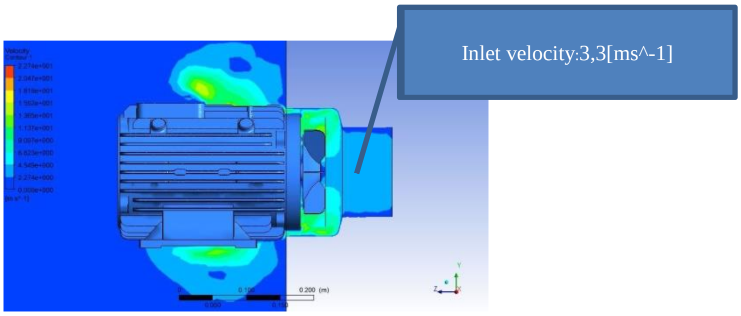

Fig. 9. Contoural View of Result of CFD for Velocity fan 1/fan cover 1

Fan 2

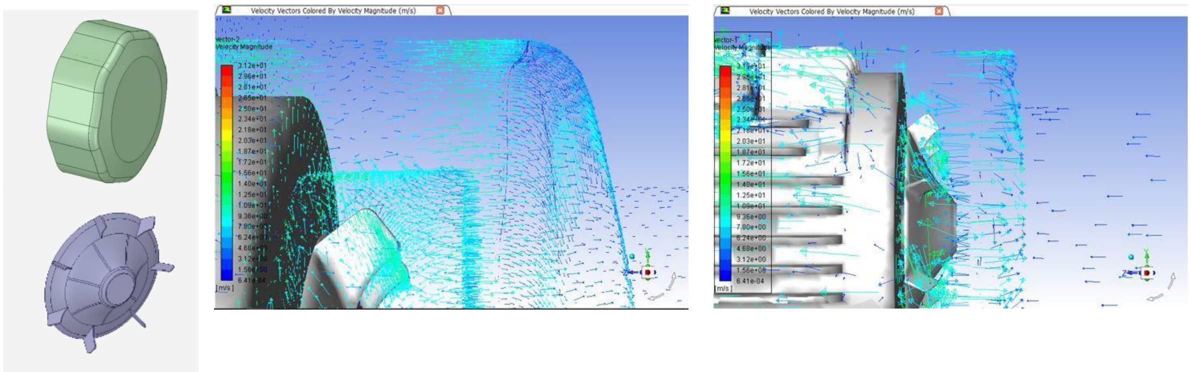

Fig. 10. Vectorial View of Result of CFD for Velocity for fan 2/fan cover 1 About Inside Fan Cover

The above analysis also shows that some of the air does not rise steeply from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door. Also, as can be seen from the colors, the flow rate is lower than the previous analysis result.

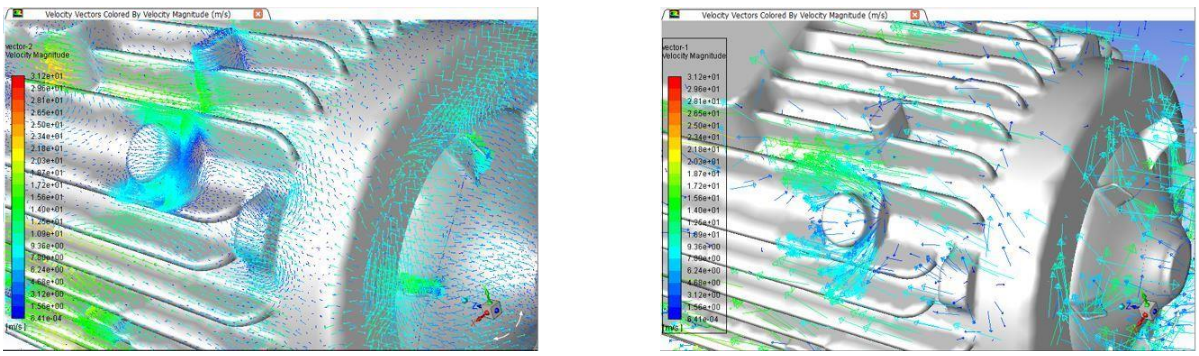

Fig. 11. Vectorial View of Result of CFD for Velocity for fan 2 / fan cover 1 Over Induction Motor Body

As a result of the above analysis, it is seen that, like the first analysis, the air has lost its velocity and direction by hitting the protrusions on the induction motor.

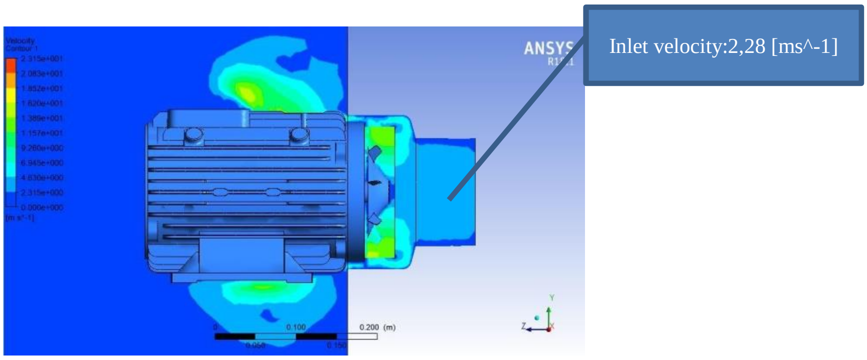

Fig. 12. Contoural View of Result of CFD for Velocity for fan 2 / fan cover 1

Fan 3

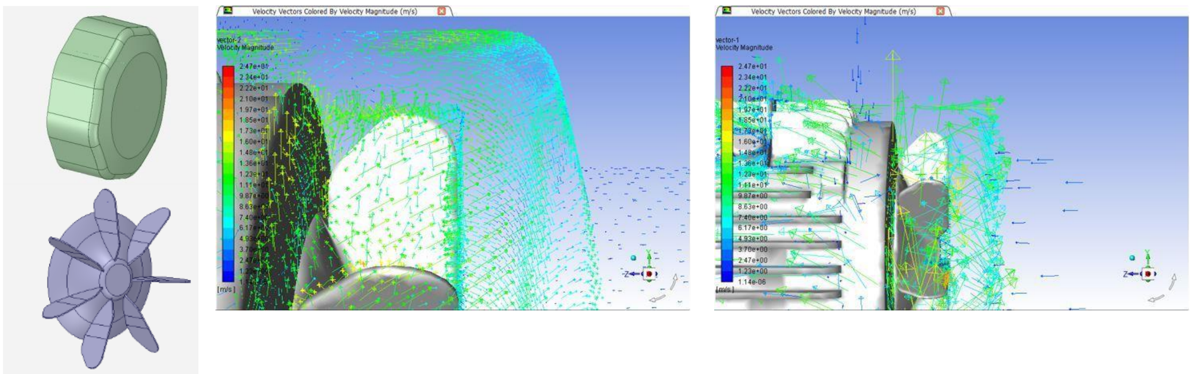

Fig. 13. Vectorial View of Result of CFD for Velocity for fan 3/fan cover 1 Over Inside Fan Cover

The above analysis also shows that some of the air does not rise steeply from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door. Also, as can be seen from the colors, the flow rate is higher than in previous analyzes.

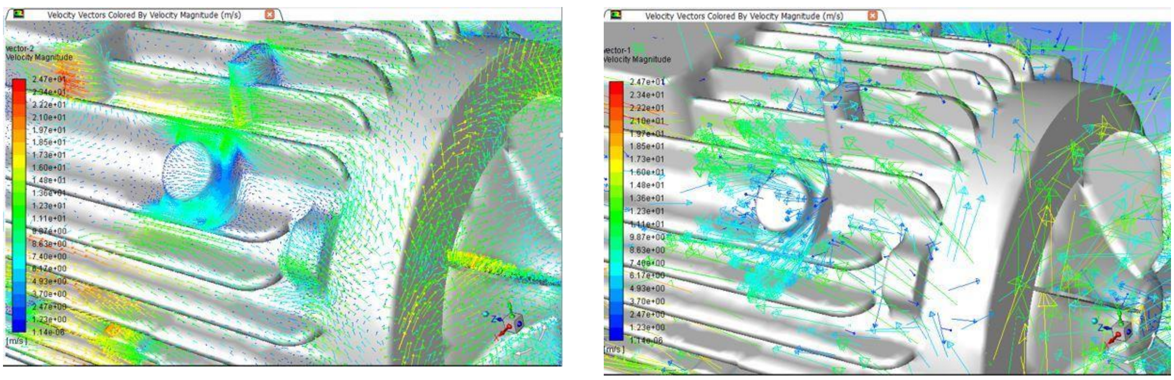

Fig. 14. Vectorial View of Result of CFD for Velocity for fan 3/fan cover 1 Over Induction Motor Body

In the above analysis, it is seen that, like the results of the previous analysis, the air has lost its velocity and direction by multiplying the protrusions on the induction motor.

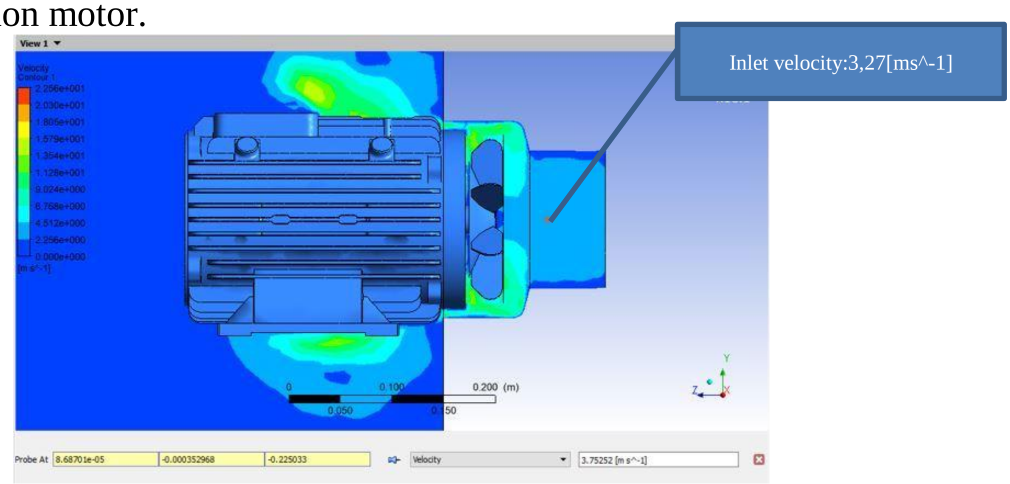

Fig. 15. Contoural View of Result of CFD for Velocity for fan 3/fan cover 1

2. Fan Cover Models
Fan Cover 1 / Fan 1

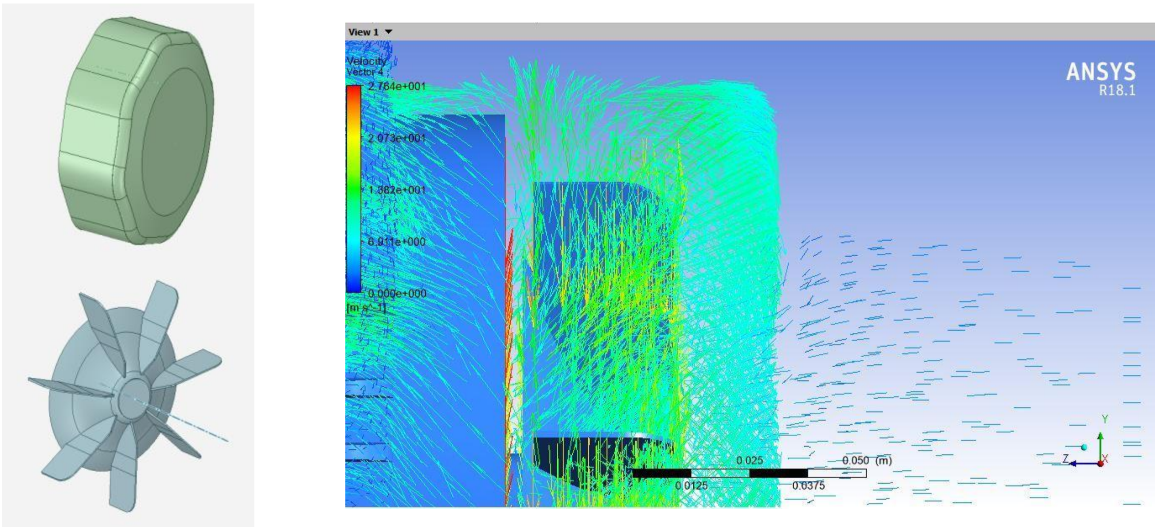

Fig. 16. Vectorial View for Result of CFD for Velocity for fan 1/fan cover 1 About Inside Fan Cover

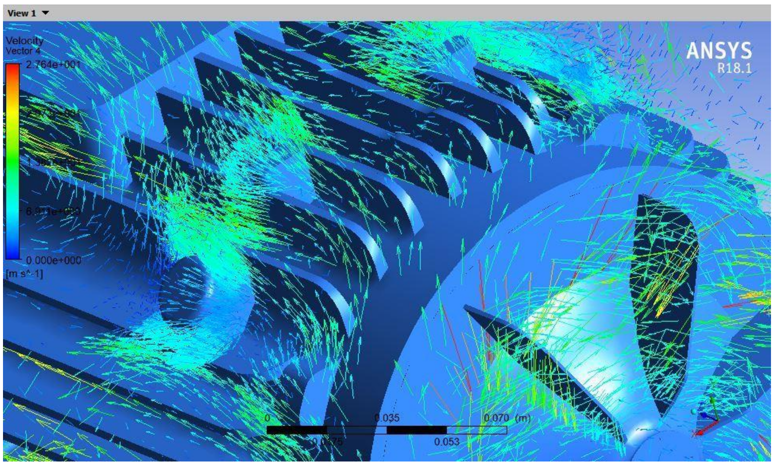

Fig. 17. Vectorial View for Result of CFD for Velocity for fan 1/fan cover 1 Over Induction Motor Body

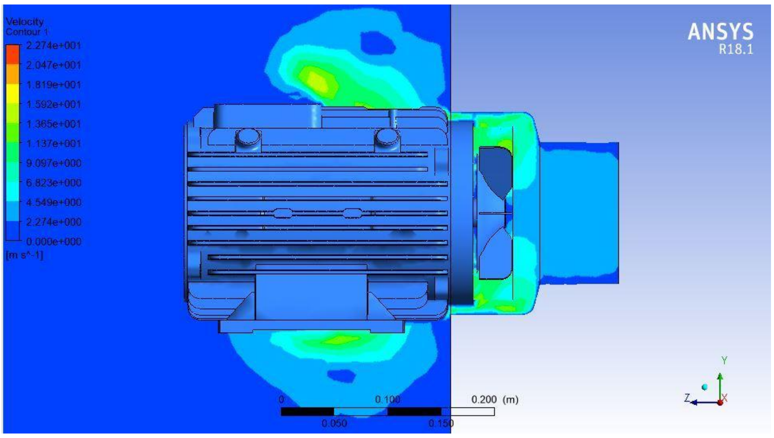

Fig. 18. Contoural Result of CFD for fan 1/fan cover 1

Fan Cover 2 / Fan 1

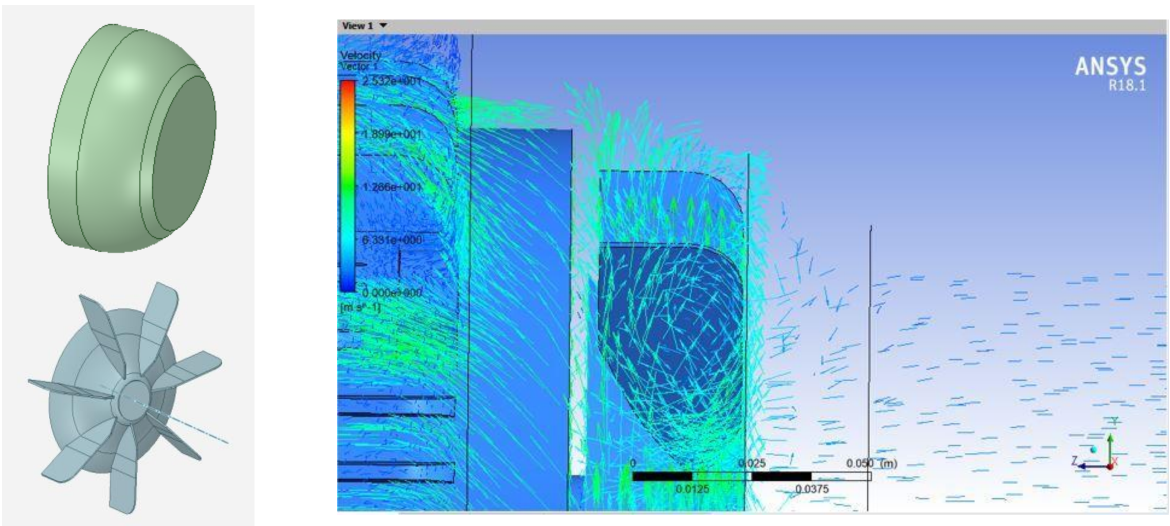

Fig. 19. Vectorial Result of CFD for fan 1 / fan cover 2 About Inside Fan Cover

The above analysis also shows that a portion of the air does not rise perpendicularly from the rotating volume. It appears that some of the convex structure on the cover allows turbulence to enter and exit, although the flow area inside the fan is less.

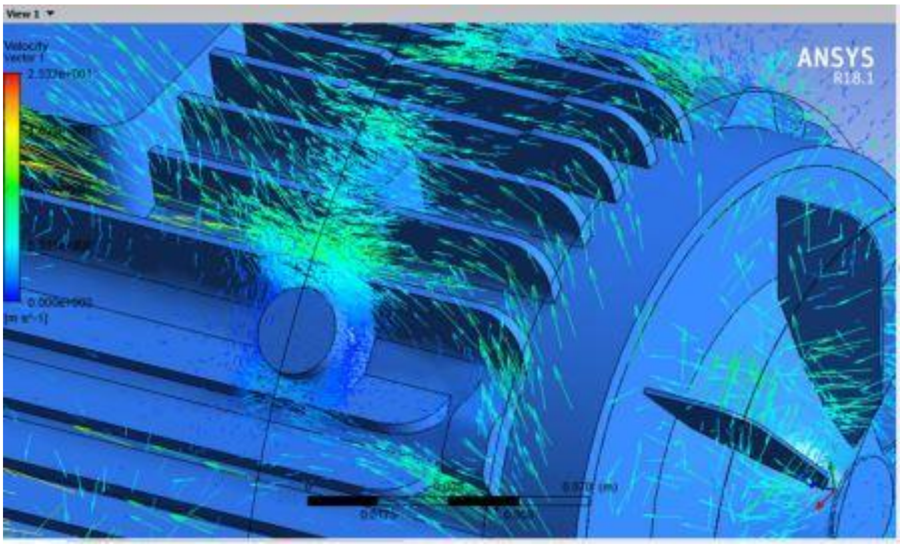

Fig. 20. Vectorial Result of CFD for fan 1/fan cover 2 About Induction Motor Body

In the above analysis, as in the previous analysis results, the air seems to have lost its velocity and direction by multiplying the protrusions on the induction motor.

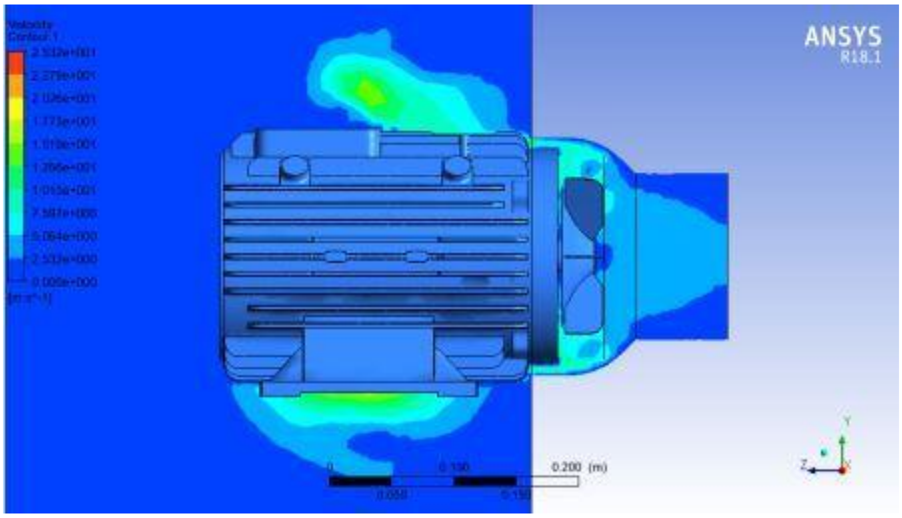

Fig. 21. Contoural View of Result of CFD for Velocity for fan 1 / fan cover 2

The fact that there is no significant change in the entrance speed indicates that the level of turbulence cannot be reduced much.

Fan Cover 3 / Fan 1

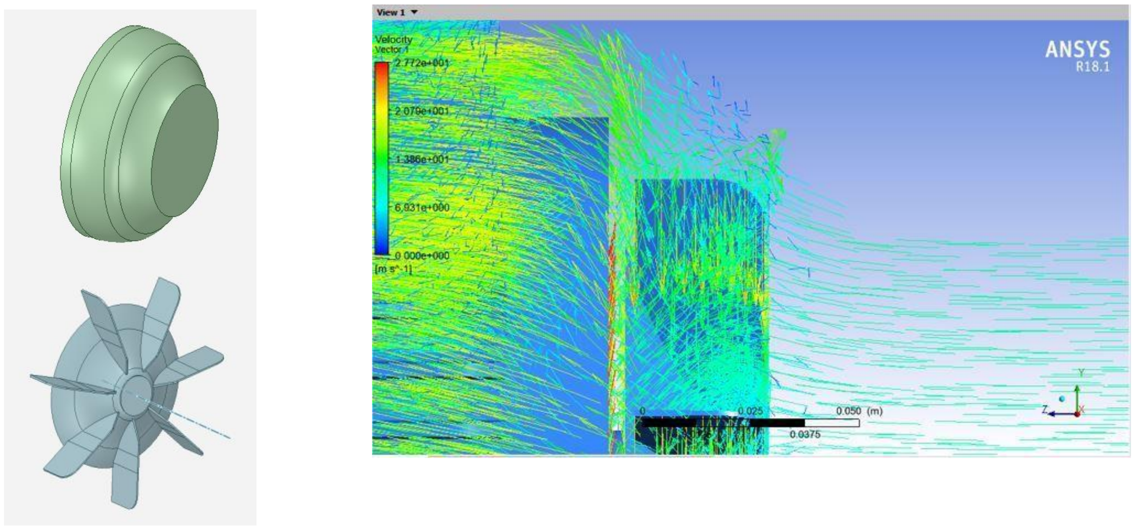

Fig. 22. Vectorial View of Result of CFD for Velocity for fan 1 / fan cover 3 about inside fan cover

As a result of the above analysis, it is seen that a very large part of the air is generated perpendicular to the rotating volume. Since both the fan left in the flow cover and the fan cover are designed in a concave shape, the air coming directly to the fan cover is directed out [6]/

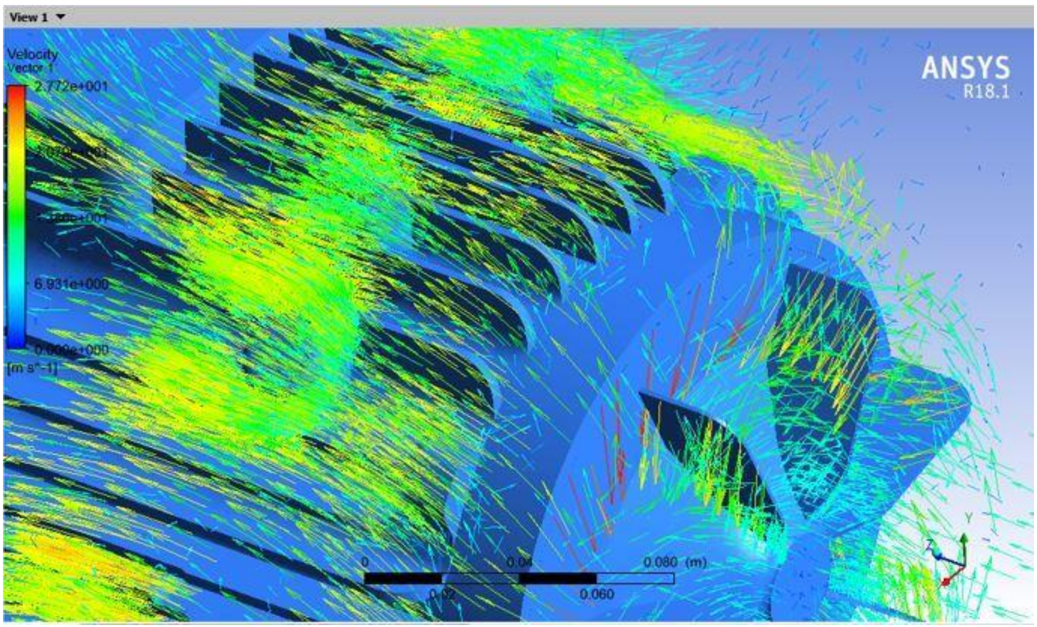

Fig. 23. Vectorial View of Result of CFD for Velocity for fan 1 / fan cover 3 about induction motor body

In the above analysis, it is seen that, like the results of the previous analysis, the air has lost its velocity and direction by multiplying the protrusions on the induction motor. As can be seen from the colors, the flow is much higher than the other analyzes.

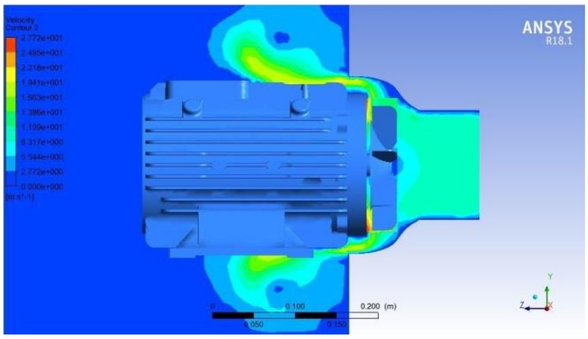

Fig. 24. Contoural View of Result of CFD for Velocity for fan 1 / fan cover 3

The fact that the turbulence level is reduced significantly indicates that the incoming air can effectively exit the fan lid.

Results of Natural Condition Experiments

To correlate the analyzes with real experiments, the velocity of inlet was used as a parameter.

Fan 1

Fig. 25. Measured Input Velocity for fan 1 / fan cover 1

Fan 2

Fig. 26. Measured Input Velocity for fan 2 / fan cover 1

Fan 3

Fig. 27. Measured Input Velocity for fan 2 / fan cover 1

Comparing of the results

Fan 1

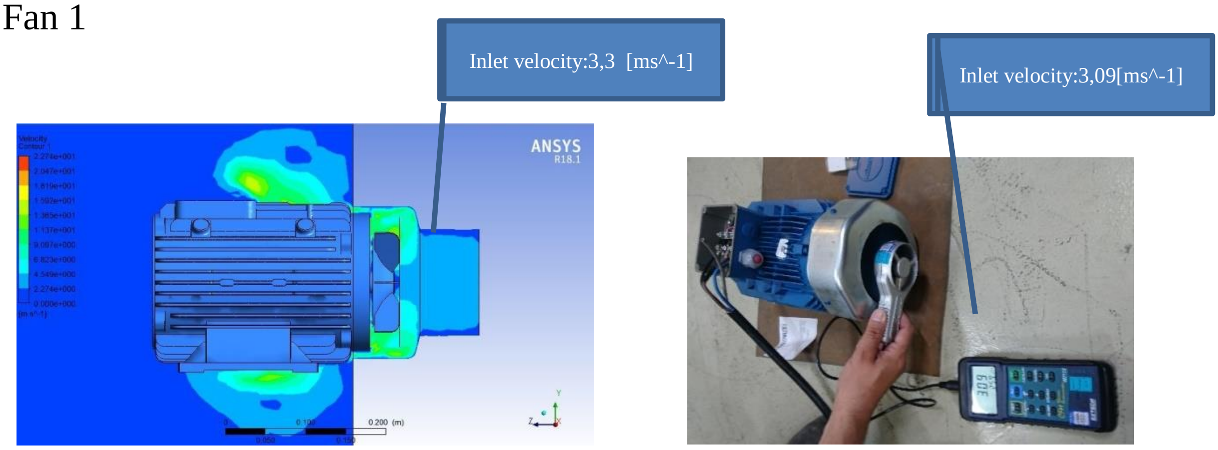

Fig. 28. Comparison of the Measured Input Velocity with The Analyzing Input Velocity for fan 1 / fan cover 1

Fan 2

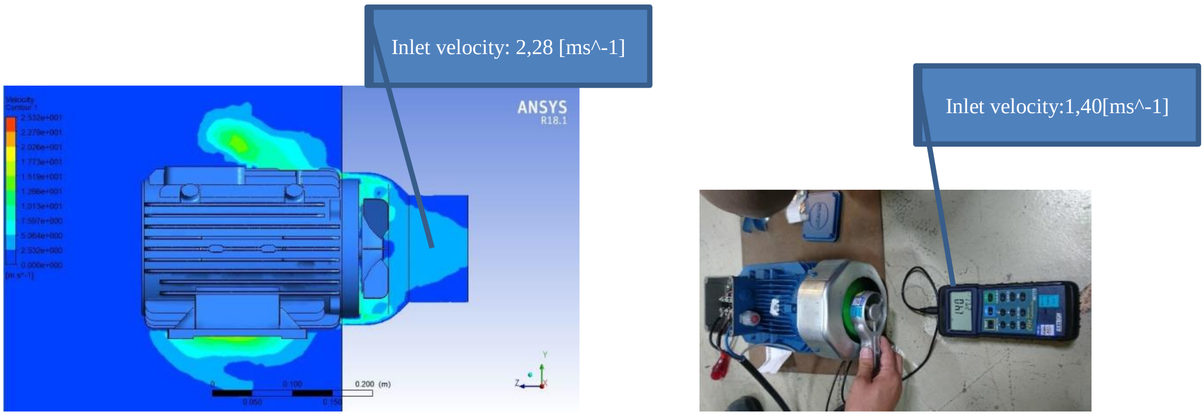

Fig. 29. Comparison of The Measured Input Velocity with The Analyzing Input Velocity for fan 2 / fan cover 1

Fan 3

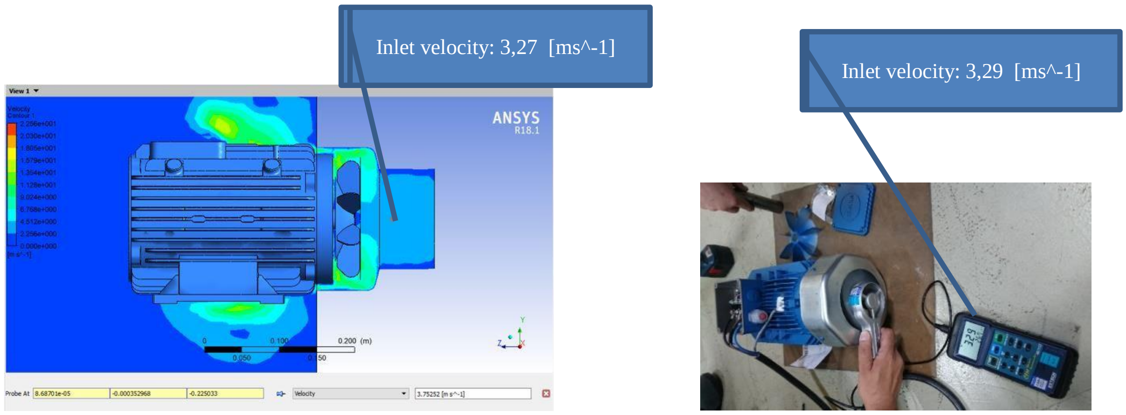

Fig. 30. Comparison of The Measured Input Velocity with The Analyzing Input Velocity for fan 3 / fan cover 1

Results and Discussion

It appears in fig. 7 – fig. 9 that some of the air does not go upright from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door. the air has lost its velocity and direction by hitting the protrusions on the induction motor.

It appears in fig. 10, at fan 2/fan cover 1, some of the air does not rise steeply from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door. Also, as can be seen from the colors, the flow rate is lower than at fan 1/fan cover 1.

It appears in fig. 11 also shows that some of the air does not rise steeply from the rotating volume. This creates a vortex effect at the edge of the fan cover. This effect reduces the speed of the flow, as well as the flow of air coming in through the fan door. Also, as can be seen from the colors, the flow rate is higher than fan 1 and fan 2.

According to the results of the first experiment, despite changing fan geometry has positive effect on the flow, the negative effect created by the vortexes has also been tested in experimental models of the fan cover designs on the elimination of the vortex formation. The following three types of fan covers in fig. 5-7 are designed for flow analysis.

It appears in the fig. 19-21 that some of the convex structure on the cover allows turbulence to enter and exit, although the flow area inside the fan is less.

As a result of the analysis, it is seen in the fig. 22 – fig. 24 that a very large part of the air is generated perpendicular to the rotating volume. Since both the fan left in the flow cover and the fan cover are designed in a concav shape, the air coming directly to the fan cover is directed out. The fact that the turbulence level is reduced significantly indicates that the incoming air can effectively exit the fan lid.

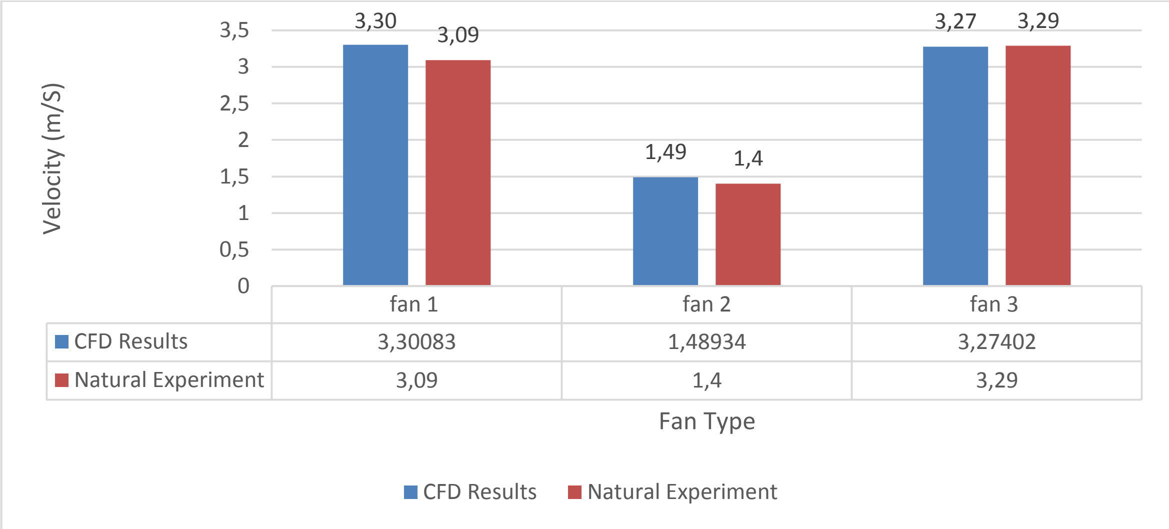

Fig. 31. Correlation between CFD results and Natural Experiment

As seen in the fig. 31, CFD results were correlated with the results of realization experiments. The realization experiments and CFD results confirm each other very high. This shows that the CFD analyzes give very close results.

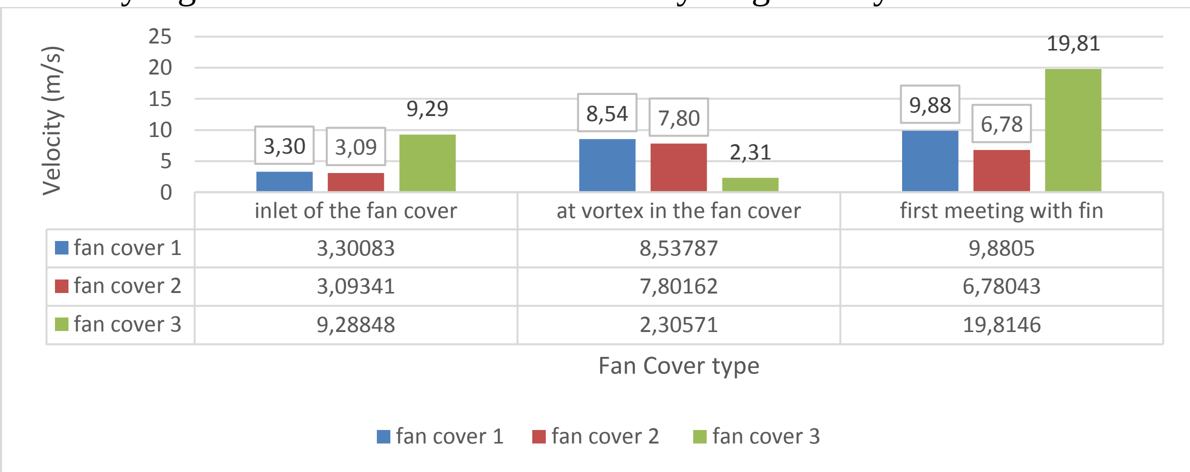

Fig. 32. Velocity of Flow According to Fan Cover Changing

As seen in the fig. 32, the realization experiments and CFD results confirm each other very high. This shows that CFD analyzes give real results.

As seen in the fig. 33, the highest vortex formation on the first cover, the highest inlet velocity and the highest first meeting with the fin velocity achieved on the third fan cover.

Conclusions

What we have seen in the results of the analysis;

The air from the fan cover entered a large turbulence in the first region that reached the fins,

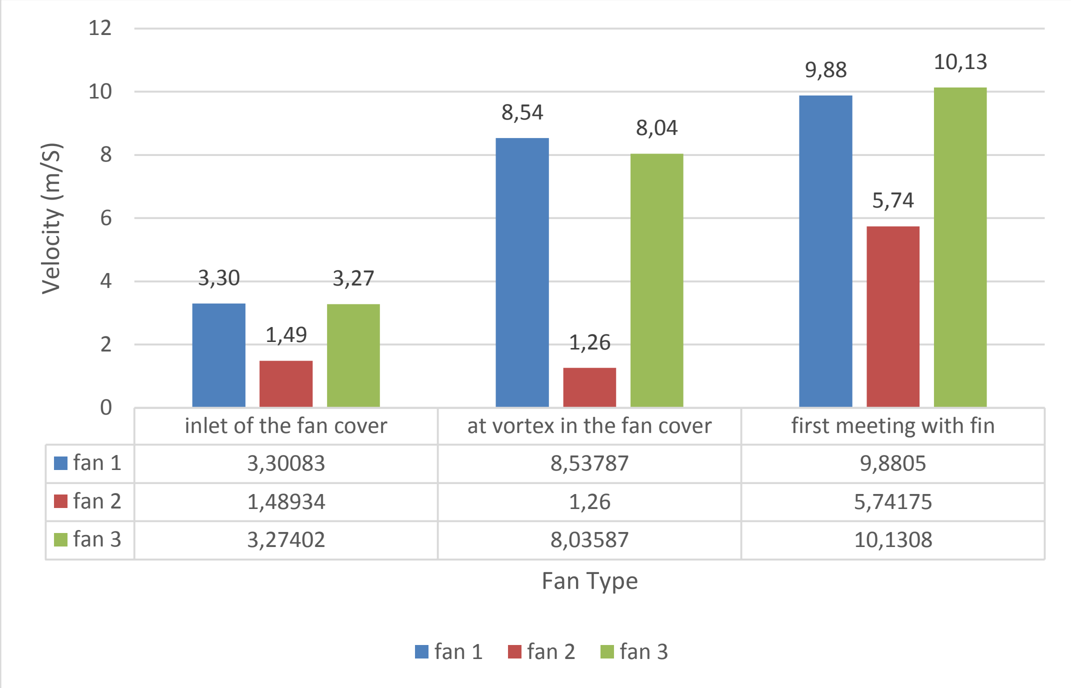

Fig. 33. Velocity of Flow According to Fan Changing

After air was removed from fan out radially, departed. (Because the radial face of the wings was flat, the flow could not know where to go and was divided into two). Some part of the flow went to the surface of the motor efficiently, and some part went in the opposite direction and broke the productive side, creating a turbulent flow there and inefficient the flow. The three designed fans showed small differences but could not solve the turbulence problem. The second of the designed fan covers could not eliminate the problem of turbulence. The third one from the designed fan covers prevented the turbulence to a large extent with its concav structure and greatly improved performance.

References

1. *Ya-Chi Chen*, *Chung-Lung Chen*, & *Qimin Dong*. (n.d.). CFD modeling for motor fan system. *IEEE International Electric Machines and Drives Conference*, 2003. IEMDC'03. doi:10.1109/iemdc.2003.1210322.
2. *Lakshminarayana B*. Fluid Dynamics And Heat Transfer Of Turbomachinery / Budugur Lakshminarayana. – Pennsylvania: A Wiley-Interscience Publication, 1996. – 809 p.
3. *Ho*, *W. H.*, & *Jermy*, *M*. Effect Of Turbulence Intensity On Vortex Formation Threshold In A Jet Engine Test Cell. *ECMS* 2011 Proceedings edited by: T. Burczynski, J. Kolodziej, A. Byrski, M. Carvalho. doi:10.7148/2011-0231-0236.
4. *Novak*, *L.*, *Bajcar*, *T.*, *Sirok*, *B.*, *Orbanic*, *A.*, & *Bizjan*, *B*. (2018). Investigation of vortex shedding from an airfoil by CFD simulation and computer-aided flow visualization. *Thermal Science*, (00), 2-2. doi:10.2298/tsci170615002n

5. *Koehler, C. M., Wischgoll, T.*, OhioLINK Electronic Theses and Dissertations Center, & Wright State University. (2010). Visualization of complex unsteady 3D flow: Flowing seed points and dynamically evolving seed curves with applications to vortex visualization in CFD simulations of ultra low Reynolds number insect flight.
6. *Gökelim, A. T.* (1983). Endüstriel fan ve kompresör tesisleri: Vantilatör ve konpresörler. İstanbul: Birsen Yayınları.