UDC 621.313.3

Gurkan Ortamevzi Ph.D., Ibrahim Emre Gunduz
(Wat Motor Industry by Arçelik Global, Turkey)
(Istanbul Technical University, Turkey)

Analysis of Cooling of Electric Motor by Radial Fan; Realization of the Experimental Analysis of CFD

It is shown by this paper, that flow analysis of fans and fan covers of different geometries are studied. The purpose of this research is to simulate the cooling state of an induction electric motor that does not yet have a prototype. Experimental results can be used for designing induction electric motors and for optimizing aerodynamic performance.

Introduction

A temperature drop of 10 degrees can extend the engine electrical insulation life by a factor of two.[1] The induction motor will run as efficiently as it can be cooled. Convection cooling is the simplest and one of the earliest techniques used[2].

Expressed as the ratio of the inlet velocity to the free flow velocity with discharge condition. This ratio is the minimum value of the Vinlet / Voutlet that the vortex can not 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 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

1) Fan Models:

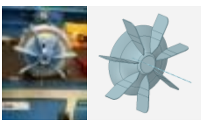

Fig. 1 *Fan 1*

Fig. 2 *Fan 2*

Fig. 3 *Fan 3*

2) Fan Cover Models

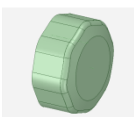

Fig. 4 *Fan cover 1*

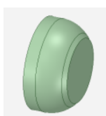

Fig. 5 *Fan cover 2*

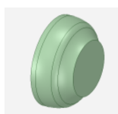

Fig. 6 *Fan cover 3*

Results of Calculation of CFD

1) Fan Models

Fan 1

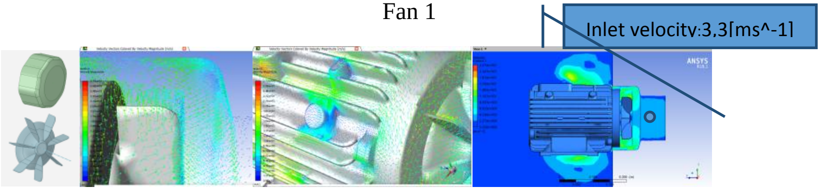

Fig. 7 *Result of CFD for Velocity for fan 1 / fan cover 1*

Fan 2

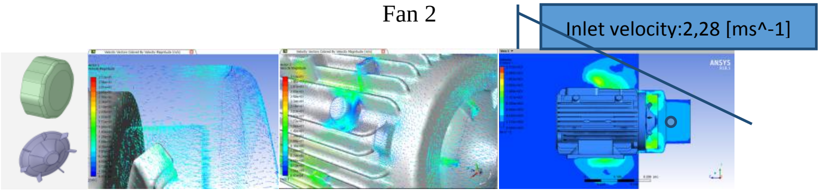

Fig. 8 *Result of CFD for Velocity for fan 2 / fan cover 1*

Fan 3

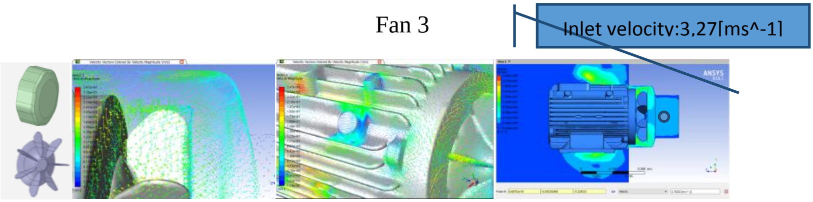

Fig. 9 *Result of CFD for Velocity for fan 3 / fan cover 1*

2) Fan Cover Models

Fan Cover 1

Fig. 10 *Result of CFD for Velocity for fan 1 / fan cover 1*

Fan Cover 2

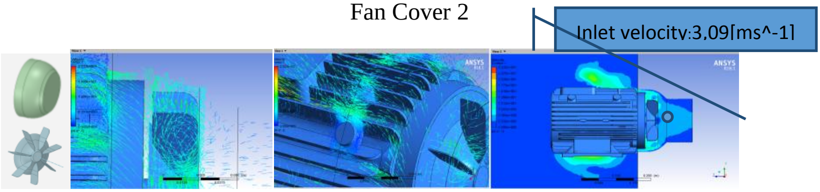

Fig. 11 *Result of CFD for Velocity fan 1 / fan cover 2*

Fan Cover 3

Inlet velocity:9,28[ms^-1]

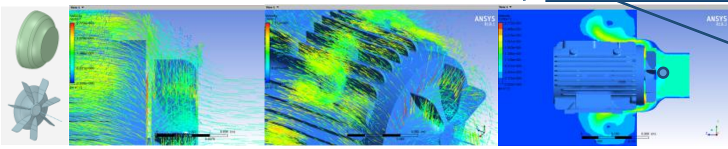

Fig. 12 *Result of CFD for Velocity for fan 1 / fan cover 3*

Results of Natural Condition Experiments

1. Comparing of the results

Inlet velocity:3,3 [ms^-1]

Fan1

Inlet velocity:3,09[ms^-1]

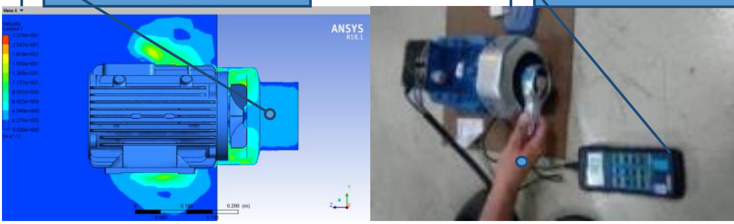

Fig. 13 *Comparison of the Measured Input Velocity with The Analyzing Input Velocity for fan 1*

Inlet velocity: 2,28 [ms^-1]

Fan2

Inlet velocity:1,40[ms^-1]

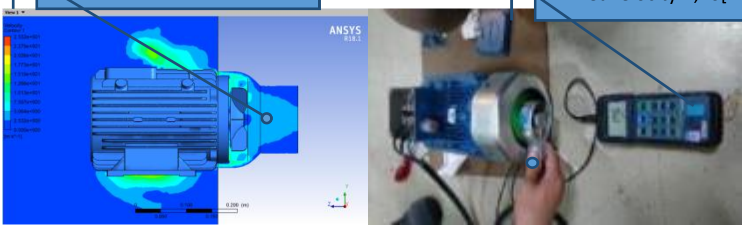

Fig. 14 *Comparison of The Measured Input Velocity with The Analyzing Input Velocity for fan 2*

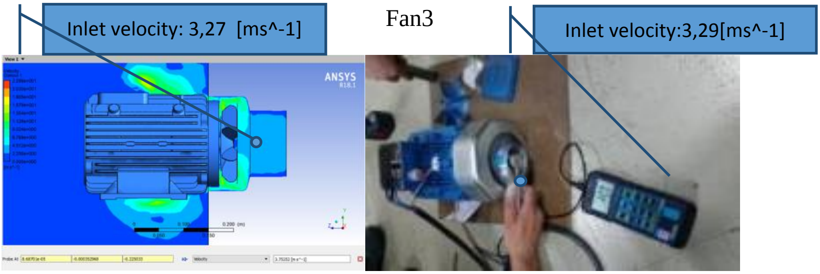

Fig. 15 *Comparison of The Measured Input Velocity with The Analyzing Input Velocity for fan 3*

2. Results and Discussion

It appears in fig. 7 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. 8, at fan 2, it is almost same. Only difference is magnitude of velocity . Also, as can be seen from the colors, the flow rate is lower than at fan 1.

It appears in fig. 9 also shows the same results. Only difference is magnitude of velocity. 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. 4-6 are designed for flow analysis.

It appears in the fig. 11 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. 12 that is seen that a very large part of the air is generated perpendicular to the rotating volume. The fact that the turbulence level is reduced significantly indicates that the incoming air can effectively exit the fan lid.

The realization experiments and CFD results confirm each other very high. This shows that the CFD analyzes give very close results.

3.Conclusions

What we have seen in the results of the analysis;

The air entered a large turbulance in the edge of fan cover. 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.) The three designed fans showed small differences but could not solve the turbulence problem. 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 с.
3. Ho, W. H., & Jermy, M. (2011). 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*.