U. SanAndres, G. Almandoz, J. Poza, G. Ugalde and A. J. Escalada, "Radial fan simulations by computational fluid dynamics and experimental validation," 2014 International Conference on Electrical Machines (ICEM), 2014, pp. 2179-2185.

DOI: https://doi.org/10.1109/ICELMACH.2014.6960486

Radial Fan Simulations by Computational Fluid Dynamics and Experimental Validation

Unai SanAndres[1], *Student Member, IEEE*, Gaizka Almandoz[1], *Member, IEEE*, Javier Poza[1], *Member, IEEE*, Gaizka Ugalde[1] *Member, IEEE*, Ana Julia Escalada[2]

[1] University of Mondragon, 20500 Arrasate – Mondragon
[2] ORONA Elevator Innovation Centre, 20120 Hernani

Φ*Abstract* – Fans are commonly used to refrigerate electrical machines when air forced cooling is required. They are rather simple solutions that can considerably improve the thermal performance of electrical machines, so the fan analysis could become an important point during the thermal design of electrical machines. This paper presents an analysis by computed fluid dynamics (CFD) simulations with the goal of characterizing radial fans, used in electrical machines. Different boundary conditions and parameters that affect to CFD results are evaluated in order to get an efficient way to simulate radial fans. The main objective is to obtain the flow-pressure characteristic curve of fans and airflow velocities by CFD simulations. Finally CFD simulation results of 2 fans are experimentally validated. One of them has been analysed in a fan testing wind tunnel. And the second one has been analysed in an auto-ventilated machine.

***Index Terms*—computational fluid dynamics (CFD), cooling systems design, fan characterization, thermal optimization, electrical machines, lumped-parameters thermal models, airflow pressure, fan testing wind tunnel, auto-ventilated machine.**

I. Introduction

FANS are widely used for electrical machines cooling when natural convection is insufficient to ensure a maximum working temperature and liquid cooling systems are too complex for specific applications. Fans impulse airflows over motor surfaces and generate forced convection as an alternative to the natural convection. So these systems can increase the external heat transfer. On the other hand, liquid cooling systems improvements are much higher than forced convection. But they need a more complex tubes system to transport the liquid, with its drive system, and a final heat exchange to transport the heat to the ambient by natural or forced air convection.

Nowadays, thermal behaviour has become a more important aspect in electrical machines even at the design stage [1]–[4] looking for smaller and more efficient machines. Since a long time ago fans have been analysed to improve the heat removal from electrical machines [5]–[8]. And in the last years, the progress in computers and computational fluid dynamics (CFD) has allowed to simulate the air motion related to electrical machines [1], [9]. Also, results from CFD can supply lumped parameter models for analytical simulations [9], [10], in order to run faster and lighter thermal simulations of electrical machines.

U. SanAndres, G. Almandoz, J. Poza, and G. Ugalde are with the University of Mondragón, 20500 Arrasate – Mondragón.
A.J. Escalada is currently with the Electrical Drives Department, ORONA Elevator Innovation Centre, 20120 Hernani.
(e-mail: usanandres@mondragon.edu; galmandoz@mondragon.edu; jpoza@mondragon.edu; gugalde@mondragon.edu; ajescalada@orona-group.com).

In the task of improving external heat transfer with cooling ducts, in [11] different shapes ducts calculating heat transfer coefficient are analysed by CFD. Whereas in [12], [13] CFD simulation models are discussed in-depth to estimate heat transfer coefficients in cooling ducts.

Interior temperatures and heat transfer coefficients are predicted in [1], [14], [15] by CFD simulations. In [1] the fluid flow in the end-region next to the end-winding is calculated to predict the convective coefficients. Whereas, rotor blades are designed as in [14] to increase the air velocity inside the machine and increase the heat transfer. And in the recent work [15] the turbulence models k-ε and SST k-ω are analysed with simulations of an axial flux machine, and validated experimentally with thermal tests.

Airflow inside machines due to shaft mounted fans is estimated in works like [16], [17] where the fan has found to consume up to 87% of total windage torque. And in [18]–[20] internal and external shaft mounted fans are simulated and validated. In [18] a wooden end-winding has been built to test an identical machine to the simulated one in CFD, by this way interior air flows are measured and validated. With the next work [19] a simplified external case with fins has been built with the same objective, to validate experimentally the simulated model in CFD. Also in these works the used mesh and models are discussed.

Regarding to fan simulations in other applications different to the motors, several works have been done to calculate or optimize fan performances. In [21] a centrifugal fan is simulated with a 2D model, whereas in [22]–[24] axial fans are simulated to refrigerate heat sinks and electronic devices. The paper [22] is a complete work of axial fan and heat sink design by CFD simulations. First, fan is designed, simulated by CFD and tested in a wind tunnel; here the 3D model, the mesh and the solvers are discussed as well. And then, an optimized heat sink is designed for this fan application. The same authors propose a parametric design to optimize fans performance in [23] modifying fan parameters and generating CFD simulations to test them.

The aim of this paper is to analyse and discuss CFD simulations looking for an efficient way to simulate radial fans, obtaining velocities and fluid flow – pressure characteristic curves. During this analysis some basic fan laws are used in order to validate CFD simulations. And finally, simulations will be also experimentally validated with a small wind tunnel and with an auto-ventilated electrical machine.

CFD simulations presented in this paper could be very useful to analyse and design cooling systems for electrical machines. So these results could be imported to finite element simulations or to analytical models like lumped parameter thermal models.

II. Fan Simulations by CFD

In this chapter, guidelines for CFD simulations of radial fans are given. These CFD simulations have been validated by experimental test, performed in two test benches: fan testing bench and an auto-ventilated machine.

The analysis performed in the current chapter is based on the fan testing bench. That is because it is necessary a system to canalize the airflow in order to measure it properly. In addition, the fan characterization consists on forcing different outlet pressure values and measuring the airflow drop. Hence, a close system is required to establish the outlet pressure, both to simulate in CFD and to test experimentally.

Therefore, measuring the fluid flow in the open channels of the auto-ventilated machine is quit more complicated so that it is not the ideal case to validate simulations.

A. *Brief introduction to fans*

Fans are used to propel air over certain surfaces to improve heat transfer due to higher air velocity, obtaining greater convective coefficients. And the value of fluid flow that a fan can achieve when it is installed in a cooling system is represented in fluid flow – pressure characteristic curves of the fan and the cooling system, as it is explained in [10].

There are different types of fans depending on the direction of the flow (axial, radial), the position of its inlet, or outlet, and the shape of the blades. Some of them are intended for high pressure applications and other ones to maximize air flows, so each one has a different characteristic curve [25]. As a very brief summary, for the same output power centrifugal fans (snail shape) generate the higher pressure with low flows, axial fans provide high fluid flows without pressure against, and radial fans are intermediate.

Similarly, when a fan is characterized fan laws provide the variation on the maximum values of mass flow, pressure, power and sound power; as consequence of the variation of blades diameter, rotation speed or air density.

B. *Simulations set-up*

First, the procedure to prepare and run CFD simulations will be explained, and after that the analysis with different simulation conditions will be addressed.

1) Geometry

The first step to prepare the simulation is the generation of the geometry of the model, taking into account that in CFD problems the model comprises the air region surrounding the solid body, and not the solid body itself (the fan). In addition it is also necessary to include additional air volumes in order to guarantee stable boundary conditions for the fluid flow. These volumes will be described later on chapter II.

The simulated fan (see Fig. 1 a)) is the typical shaft mounted fan used in auto-ventilated machines. It comprises 7 blades, the hub diameter is 50 mm and external diameter is 165 mm. And the considered air region will be volume enclosed between the external case and the fan as can be seen in Fig. 1 b) and c).

In the model it is necessary to name surfaces to configure specifically the mesh, boundary conditions and to get results at these surfaces.

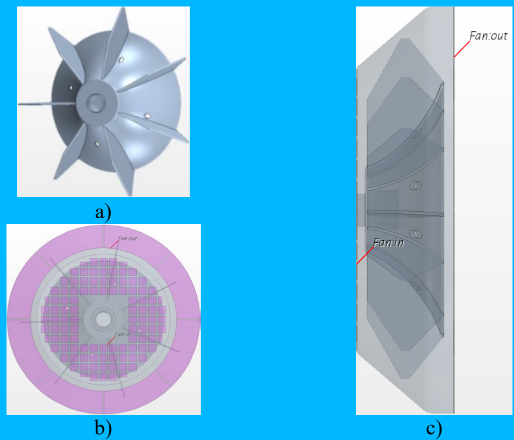


Fig. 1. a) Fan to characterize and air model for the fan: b) front view, c) side view

2) Boundary Conditions

Boundary conditions for simulations define the behaviour of the flow at surfaces. These boundary conditions must be defined as: inlet or outlet of the domain to define a fluid exchange with the ambient, static walls where the fluid cannot penetrate, periodic faces if the model is not complete and results are repetitive, and interfaces to simulate fluid exchange between bodies in case more than one bodies are simulated.

In these simulations the outlet will be set as *Outlet Pressure*, and the inlet as *Stagnation Inlet* (later *Mass Flow Inlet* will be analysed as well). When periodicities are used these faces will be defined as *Rotational Periodic*. Static walls around the fan will be set with *Tangential Velocity: None Absolute*, whereas the rest of the surfaces will be *Tangential Velocity: None Relative to Mesh*, in order to rotate with the fan.

3) Mesh

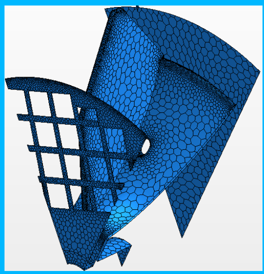
The mesh of the fan model is based on *Polyhedrals*, and the base size for the elements is 3 mm, with a minimum size of 10% (0.3 mm). When the overall fan testing tunnel is simulated, in this region the elements are 10 times bigger in order to reduce the mesh total size. And if the inlet and outlet spheres for boundaries are included, the exterior elements will be 100 times bigger to distance the domains limits without exceeding the number of elements.

Regarding to the mesh quality, the difference on the size of the elements between near surfaces is limited to 10 times. Because for the same surface the smaller elements are at least the 10% than the biggest ones and in boundary spheres the bigger elements are only at exterior surfaces. So checking the maximum value of *skewness* in the mesh it is between 80 and 85º (maximum recommended value). Also, the wall y+ parameter which relates the fluid velocity near the wall with the mesh element size, it is below 100 and the average value is about 30 as it is recommended.

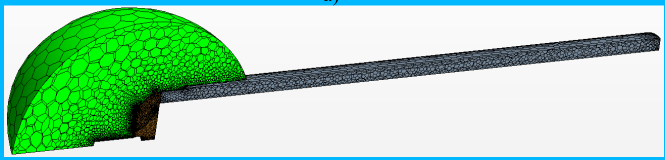
4) Solve

The air volume around the fan is configured to rotate at 1500 rpm, and the solver used is *Realizable K-Epsilon Two Layer All y+ Wall Treatment*, with the fluid set as air with constant density. This solver runs *K-Epsilon* model taking into account the air velocity and mesh size near walls.

During the fan characterization the pressure value set at the inlet is 0 Pa, whereas at the outlet several simulations are run with different values of static pressure to complete the characteristic curve.



a)



b)

Fig. 2. Mesh of the models: a) air surrounding the fan, b) with air sphere for inlet boundary and tunnel

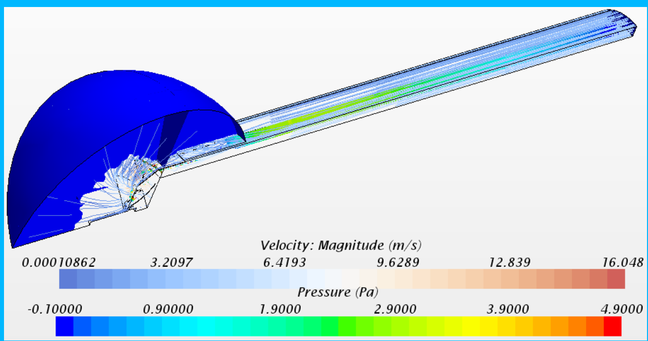


Fig. 3. Velocity and Pressure results for free airflow in the tunnel (1/7 periodicity)

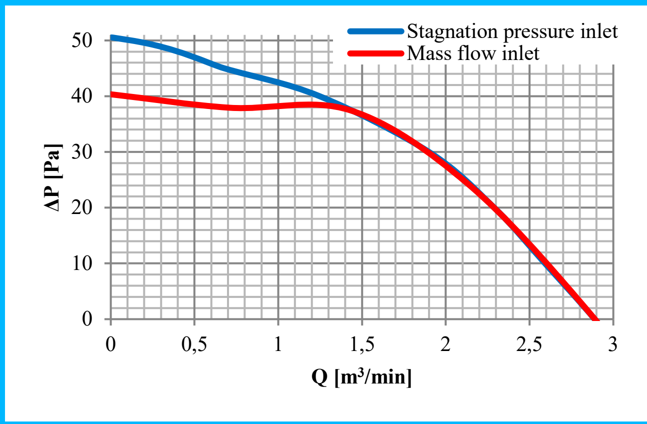


Fig. 4. Inlet boundary conditions comparison: *Stagnation Pressure inlet* vs *Mass flow inlet*

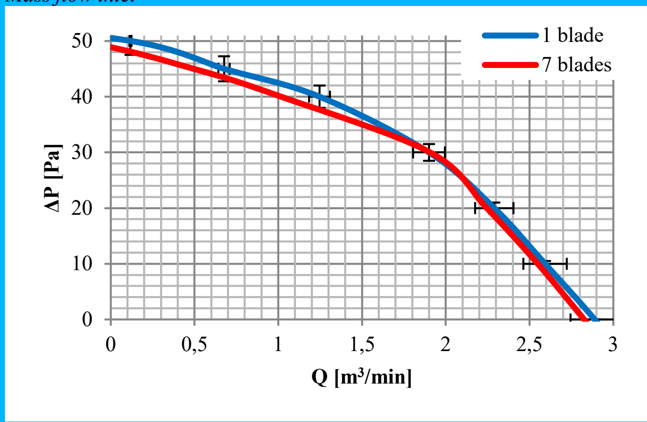


Fig. 5. Periodicities analysis: 1 blade model vs 7 blades complete model

5) Results

The characteristic curve of the fan is the result of various simulations, imposing different static pressures at the outlet. Each simulation is a point over the characteristic curve of the fan shown in the Fig. 5. This value of pressure is the pressure at the outlet of the tunnel and the fluid flow is obtained with the air density and the mass flow at the inlet or the outlet (they must be equal for a valid simulation).

At the end of every simulation the value of iteration residuals must be lower than 10^{-4} to ensure a stable result during iterations, and the pressure and mass flow results must be stable too. Besides, it is important to check the pressure in the domain and streamlines of air velocity to validate the simulation. In Fig. 3 can be shown the fan in the wind tunnel with 0 Pa of static pressure at the outlet, it would represent free airflow from the fan across the tunnel.

C. Analysis of simulation parameters

Once the procedure to characterize the fan by CFD simulations has been defined, some simulations configurations and parameters will be analysed to study their influence on the results and to find the best option.

1) Boundary types

Boundaries are the surfaces that limit the bodies as has been explained before boundary conditions indicate the fluid behavior near these surfaces. Walls represent surfaces where fluid cannot cross, periodical faces repeat the results periodically at them and inlet and outlets allow a fluid exchange with the ambient. It can be defined as a value of pressure for the external environment (*Stagnation Inlet*, *Pressure Outlet*) or fluid flow (*Velocity Inlet* for incompressible flows, *Mass flow inlet* for any flow).

It is recommended by CFD software user manuals (*Star-CCM+*, Fluent) and typical to simulate CFD problems, to set the outlet as *Pressure Outlet*. However for the inlet boundary condition there are two main options: defining the fluid flow with *Velocity inlet* or *Mass flow inlet*, and defining an ambient pressure with *Stagnation pressure inlet*. The first parameter to analyse will be these inlet boundary conditions as it is analysed in [10].

The same model of fan has been simulated to analyse the characterization obtained with each boundary condition. As can be seen in Fig. 4, when the air flow is imposed with the boundary *Mass flow inlet* the airflow is lower at high values of static pressure at the outlet.

2) Periodic boundary condition

Periodic surface is a type of boundary condition involving repeat results from one surface to another one. This reduces the size of the model which implies lower memory requirements, lower computational load and lower computing time. Simulating fans the periodicity of the model corresponds to the number of blades, so it could be possible to simulate a single fan blade as it is shown in Fig. 7 a).

The results obtained simulating only 1 blade or the complete model comprising 7 blades, are represented in Fig. 5. Differences in results of these two models are lower than 5% which can be considered as acceptable.

3) Boundary air volumes surrounding the fan

The definition of the boundary conditions requires a stable fluid flow at those surfaces. In addition it needs to be as perpendicular as possible and it is recommended to avoid reverse flows. Therefore, boundary surfaces must be far enough away from fan blades where pressure is unstable.

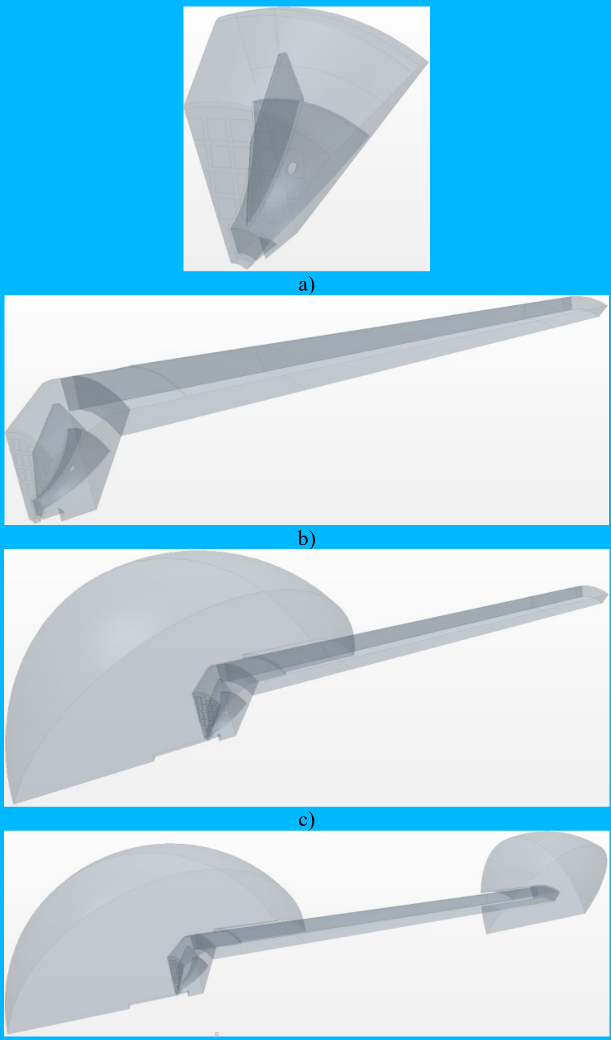


Fig. 6. Boundary air volumes to characterize the fan: a) only air enclosed between fan and cap, b) tunnel, c) inlet sphere, d) outlet sphere

In addition, for the fan characterization it is necessary to include the air volume surrounding the fan; that in this case is assumed to be enclosed by its cap. In order to analyse the influence of these boundary air volumes and the stability of boundary conditions 4 different simulations have been carried out (see Fig. 6). In the first one, only the air volume enclosed between the fan and the cap is considered. In the second one, a fan testing wind tunnel is added to measure a stable fluid flow at the output as it will be done during experimental validation. For the third model, an input sphere is used to stabilize the air in the fan inlet and ensure the ambient pressure. Finally in the fourth simulation an outlet sphere intends to do the same work for the outlet.

The results are represented in Fig. 7. Looking at these results it could be stated that the inlet sphere becomes necessary to assure a stable ambient at the inlet and also to measure a correct pressure difference. So the first two options are dismissed. On the other hand, the tunnel is required to direct the flow and set the outlet pressure. Regarding to the outlet sphere, it affects only to the maximum static pressure value.

4) Air Density

The density of the air affects to the experimental validation because it depends on the altitude, temperature and humidity. And it affects also to the value of maximum pressure that the fan can reach.

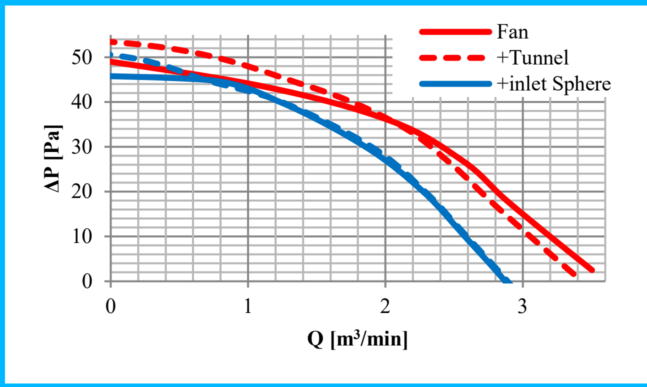


Fig. 7. Boundary air volumes influence on fan characterization

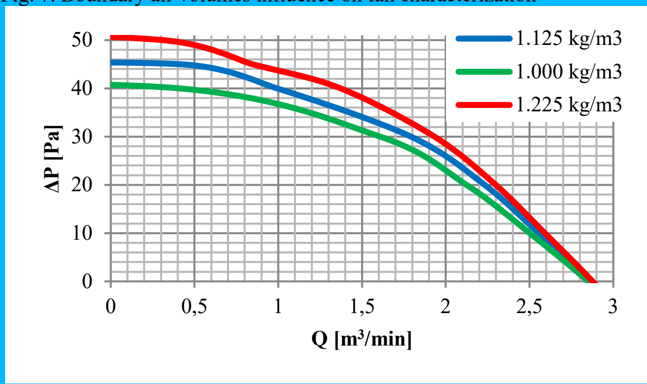


Fig. 8. Air density influence on air volume flow

TABLE I

FAN LAWS: VARIATION ON AIRFLOW AND STATIC PRESSURE DUE TO DIAMETER, SPEED AND AIR DENSITY

	Airflow [m^3/s]	Pressure[Pa]
Diameter	$q_v = q_{v0}\left(\frac{D_r}{D_{r0}}\right)^3$	$P_F = P_{F0}\left(\frac{D_r}{D_{r0}}\right)^2$
Speed	$q_v = q_{v0}\left(\frac{n}{n_0}\right)$	$P_F = P_{F0}\left(\frac{n}{n_0}\right)^2$
Air Density	$q_v = q_{v0}$	$P_F = P_{F0}\left(\frac{\rho}{\rho_0}\right)$

Fan laws expressed in TABLE I provide the variation that fan diameter, rotating speed and air density produce on static pressure and fluid flow. These equations can be used to estimate a new fan characteristic curve when any of these parameters has changed.

The influence of the air density on CFD results has been validated changing the value in the simulation and getting new characteristic curves. As can be seen in Fig. 8, with higher values of air density the pressure value increases proportionally to it. The maximum volume flow remains constant whereas a higher air density implies that the fan produces a higher pressure due to the higher air mass in the same air volume. So CFD results are consistent with fan laws of TABLE I.

5) Rotational speed

As mentioned before the rotational speed modifies the fan characteristic curve. The air flow increases proportional to the increased speed, and the static pressure increases as the square. Validation of CFD simulations for different rotational speeds is exposed in Fig. 12.

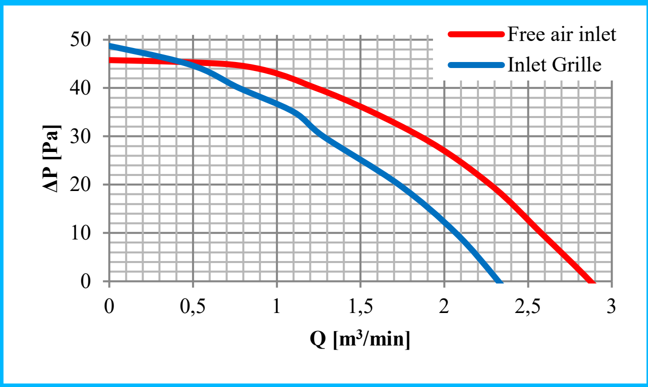


Fig. 9. Inlet fan cover grille reduces the maximum airflow about 20%

In that case the fan has been simulated at different speeds, but it should be mentioned that simulating the fan for a single speed would be enough. And after that the fan performances at other speeds could be deduced applying the basic fan laws.

The fan law of the TABLE I, related to rotating speed expresses the variation on the fan characteristic curves. In Fig. 12 it is compared the characteristic curve when the fan is rotating at 1500 rpm and the estimations to lower speeds, with CFD simulations changing the rotating speeds. And here again CFD results are consistent with fan laws.

6) Inlet grille

Finally, the fan cover grille (see Fig. 11) that is present in the fan cap and merges with the tunnel is included in the model to study the decrease of fluid flow. In Fig. 9 a reduction of about 20% in the maximum air flow value can be seen. So the consideration of that element is essential in order to achieve enough accurate results.

III. Case Studies

During the previous chapter the set-up to simulate radial fans in CFD has been explained. In addition the influence of some boundary conditions and parameters have been analysed with the aim of defining a procedure to characterize fans by CFD.

In that chapter CFD simulations will be validated experimentally considering two case studies. In the first one, a Siemens radial fan has been tested in the fan testing tunnel to obtain the full characteristic curve. In the second case study, the same fan has been mounted in the shaft of a Siemens machine. In this second test bench the air velocity along cap ducts has been measured at rated speed of the motor.

A. Fan Characterization in fan testing Wind Tunnel

The fan testing tunnel is the best way to characterize fans because the fan can be tested against different outlet pressures in order to obtain the corresponding flows. This way the flow-pressure characteristic curve can be plotted.

1) Description of the Test Bench

A fan testing tunnel to test radial fans has been built which is shown in Fig. 10. It comprises two cylinders, inner and outer in order to canalize the airflow between them (see Fig. 11). The outer one has a 180 mm diameter. Both cylinders are separated by 8 ducts to set right the air flow reducing rotational components. The inner cylinder has a 150 mm diameter, and it is necessary to prevent reversed flows. In fact this system is emulating the housing of the electrical machine which is the application that the fan is designed for.

Fig. 10. Testing wind tunnel for radial fans

a) b)

Fig. 11. Detail of the wind tunnel: a) fan cover, b) front view of the fan with the tunnel

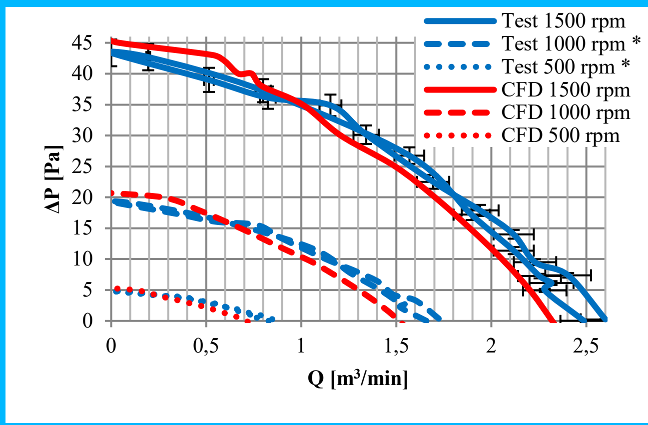


Fig. 12. Comparison of the characteristic curve obtained experimentally at 1500 rpm, estimations with fan laws at lower speeds and curves simulated

2) Fan characterization

The method to characterize the fan consists on rotating the fan at required speed and blocking the outlet gradually measuring the current airflow and the static pressure near the outlet.

In fact the measured variables are not completely equal in all the 8 ducts. Due to that for every valve position, individual measurements are required in each duct for airflow and static pressure. In addition the airflow distribution inside ducts is not uniform and the measured value depends on the orientation of the probe. Considering all these details the maximum values of airflow has been logged inside every duct.

In Fig. 12 the obtained results when the fan is rotating at 1500 rpm are represented. It is important to notice that each point represents the average value of airflow and pressure computed considering the measurements taken in all ducts. Two curves relate the cycle of closing and opening the output valve. The test was performed in Arrasate-Mondragon (Gipuzkoa) on November 12th and the air density was estimated to be about 1.15 kg/m^3.

3) Experimental validation at different rotating speeds

The test has been run at rated speed 1500 rpm to reduce the measurement errors because the higher the speed is the higher the measured magnitudes are and so the higher the accuracy is. Fan characteristics curves for lower speeds have been deduced applying the basic fans laws.

In Fig. 12 CFD results for different rotational speeds are compared to experimental measurements. In the test curve at 1500 rpm 5% error bars have been superposed to analyse results. It can be seen that the error is lower than 10%.

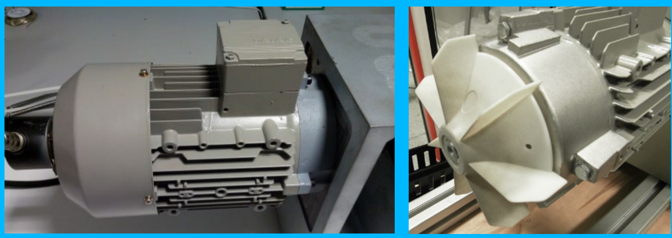
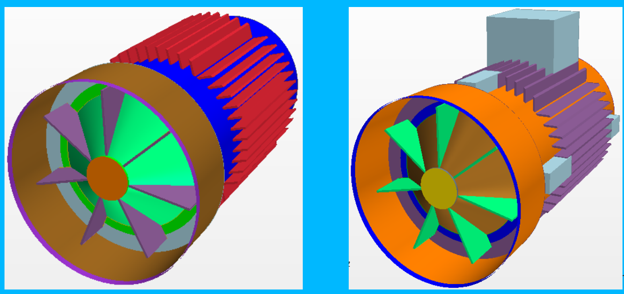


Fig. 13. a) Auto-ventilated Siemens machine to test and b) detail of the shaft mounted fan without the fan cover



a) b)

Fig. 14. Simulated Fan models within Siemens machine: a) 3D model generated in *Motor-CAD* and imported to *Star-CCM+*, and b) modified with connection box and cover screws

B. *Auto-Ventilated machine*

The next case study to validate CFD fan simulations is a shaft mounted fan in a Siemens machine that can be seen in Fig. 14. The final application of this work is to simulate fans in electrical machines obtaining information for later works like thermal simulations.

1) *Model generation*

The 3D model to simulate in CFD has been created with the thermal simulation software for electrical machines *Motor-CAD*. It has an export application to generate 3D models directly to *Star-CCM+* (see Fig. 15). So in *Motor-CAD* the exterior geometry of the electrical machine has been defined: the external cap, fins, the fan and its cover.

The fan is similar to the one which has been validated in previous chapters but it has some differences in the shape of the blades as can be seen in Fig. 14 b). On the other hand, *Motor-CAD* is an analytical software so that it is not possible to model all details of the geometry Therfore the simulated 3D model has some small differences with the real machine. Regarding to that, after the import operation in *Star-CCM+* some modifications have been done to include the connections box and some screws on the cap. The comparison of the model imported from *Motor-CAD* and the modified can be seen in Fig. 15 a) and b), also their results are in Fig. 17.

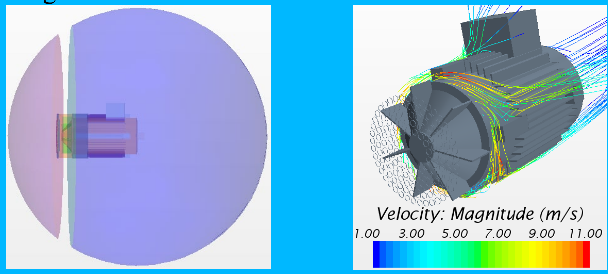
2) *CFD simulation*

In that particular case the aforementioned boundary air volume is a sphere surrounding the overall electrical machine. This sphere is divided in two parts generating inlet and outlet surfaces (see Fig. 16 a)).

The mesh is polyhedral based and there are not prism layers. Nevertheless Some simulations with prims layers have been also carried out but not big differences have benn found when wall y+ is maintained below 50. The default size of the elements has been set to 2.5 mm for fan zones and external fins, and 10 mm for the rest of the region. The total size of the mesh is about 500,000 elements.

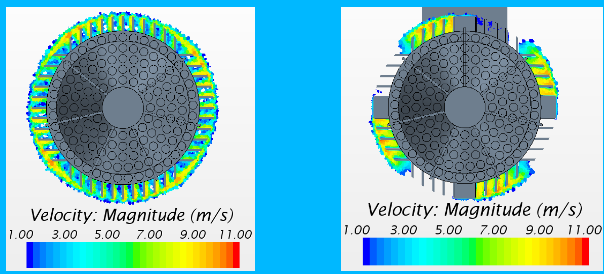
3) *Experimental validation at different rotating speeds*

CFD simulations of the shaft mounted fan shown in Fig. 14, have been validated at three different rotating speeds. And the air flow has been measured using a pitot tube in all cooling ducts at the outlet of the cover.



a) b)

Fig. 15. a) Boundary air volumes and b) airflow streamlines at 1500 rpm



a) b)

Fig. 16. Air velocity on the outlet of the fan cover in: a) imported model from *Motor-CAD* and b) modified model with connection box and screws

As it can be seen in Fig. 17 when connection box and cover screws are added to the problem there is a significant shadow effect behind them, so that in some channels the air flow is practically nul depending on the rotation direction of the fan. So it has been simulated and analysed experimentally.

In experimental tests, the maximum measured airflow in the outlet of the fan cover has been about 11 m/s at the exterior zone, similar to CFD simulations. In ducts with perpendicular fins the measured air velocity has been between 5.1 m/s and 9.3 m/s, which also agrees with CFD results. Regarding to the ducts with lower speeds, experimentally 3 ducts with zero or reverse air flow have been found. So it agrees with CFD simulations in which screws and connexion box have been taken into account. But in the next two ducts farthest from the shadow zone, 8 m/s and 6.6 m/s have been measured disagreeing with CFD simulations. In addition, just in the middle of the axial length of the machine the measured maximum air velocities have been about 7.7 m/s in concordance with the simulation results. And there, at that distance from fan cover, it has been proved that the airflow is more stable. It has been also validated that the higher airflow is close to the motor surface as predicted in simulations.

For 1000 rpm and 500 rpm rotating speeds, the airflow behaviour is the same. The measured maximum airflow in the outlet of the fan cover has been about 6.5 m/s and 3.2 m/s, respectively. The error between CFD results and experimental measurements has been lower than 10%.

IV. Conclusions

In this paper, radial fan simulations by CFD are discussed in order to define the best way to carry out these simulations. The model generation has been explained and simulations with different boundary conditions have been carried out to compare results. And finally, CFD simulations have been validated with 2 case studies: fan testing wind tunnel to characterize radial fans, and an auto-ventilated machine to measure the airflow created by the shaft mounted radial fan.

In the CFD simulations analysis it has been found that inlet pressure with outlet pressure conditions is the best way to characterize fans. In addition periodic condition can be used to reduce memory requirements and CPU load, as it was concluded in [10]. Regarding to boundary air volumes, an inlet sphere is necessary to make stable the inlet boundary condition and also the wind tunnel to measure airflow and static pressure, which will give the fan characteristic curve. According to fan laws, an air density increase affects making higher the static pressure value maintaining the airflow, and the rotating speed increase proportionally the airflow and the static pressure squared. Finally, the grille in the inlet of the cover fan reduces about 20% the maximum airflow, so it has to be included in simulations.

Finally, a radial fan has been characterized by CFD obtaining the characteristic airflow – pressure curve and has been validated experimentally in wind tunnel with an error below 10%.

On the other hand, a second radial fan mounted in an auto-ventilated motor has been simulated by CFD and tested experimentally at different rotating speeds. The average error of the airflow measurements is about 10%, and the effect of the cover obstacles has been analysed.

V. References

[1] D. A. Staton, S. . Pickering, and D. Lampard, "Recent Advancement in the Thermal Design of Electric Motors," in *2001 SMMA Fall Technical Conference "Emerging Technologies for the Electric Motion Industry*, 2001.

[2] K. Hruska, V. Kindl, and R. Pechanek, "Concept, design and coupled electro-thermal analysis of new hybrid drive vehicle for public transport," in *14th International Power Electronics and Motion Control Conference (EPE/PEMC)*, 2010, pp. S4–5–S4–8.

[3] A. Boglietti, A. Cavagnino, M. Popescu, and D. Staton, "Thermal Model and Analysis of Wound-Rotor Induction Machine," *IEEE Trans. Ind. Appl.*, vol. 49, no. 5, pp. 2078–2085, 2013.

[4] M. Hafner, M. Schoning, and K. Hameyer, "Automated sizing of permanent magnet synchronous machines with respect to electromagnetic and thermal aspects," in *18th International Conference on Electrical Machines. ICEM 2008.*, 2008, pp. 1–6.

[5] D. B. Hoseason, "The cooling of electrical machines," *Electrical Engineers, Journal of the Institution of*, vol. 69, no. 409. pp. 121–143, 1931.

[6] R. Poole, "The application of propeller fans to the cooling of electrical machines," *Electrical Engineers, Journal of the Institution of*, vol. 77, no. 465. pp. 293–304, 1935.

[7] S. Noda, S. Mizuno, T. Koyama, and S. Shiraishi, "Development of a totally enclosed fan cooled traction motor," *2010 IEEE Energy Convers. Congr. Expo.*, pp. 272–277, Sep. 2010.

[8] S. Mizuno, S. Noda, M. Matsushita, T. Koyama, and S. Shiraishi, "Development of a Totally Enclosed Fan-Cooled Traction Motor," *Industry Applications, IEEE Transactions on*, vol. 49, no. 4. pp. 1508–1514, 2013.

[9] A. Boglietti, A. Cavagnino, D. Staton, M. Shanel, M. Mueller, and C. Mejuto, "Evolution and Modern Approaches for Thermal Analysis of Electrical Machines," *IEEE Trans. Ind. Electron.*, vol. 56, no. 3, pp. 871–882, 2009.

[10] U. SanAndres, G. Almandoz, J. Poza, and G. Ugalde, "Design of Cooling Systems Using Computational Fluid Dynamics and Analytical Thermal Models," *Industrial Electronics, IEEE Transactions on*, vol. 61, no. 8. pp. 4383–4391, 2014.

[11] Z. Huang, F. Marquez, M. Alakula, and J. Yuan, "Characterization and application of forced cooling channels for traction motors in HEVs," *Electrical Machines (ICEM), 2012 XXth International Conference on*. pp. 1212–1218, 2012.

[12] M. Schrittwieser, A. Marn, E. Farnleitner, and G. Kastner, "Numerical analysis of heat transfer and flow of stator duct models," *Electrical Machines (ICEM), 2012 XXth International Conference on*. pp. 385–390, 2012.

[13] M. Schrittwieser, A. Marn, E. Farnleitner, and G. Kastner, "Numerical Analysis of Heat Transfer and Flow of Stator Duct Models," *Industry Applications, IEEE Transactions on*, vol. 50, no. 1. pp. 226–233, 2014.

[14] C. Jungreuthmayer, T. Bauml, O. Winter, M. Ganchev, H. Kapeller, A. Haumer, and C. Kral, "A Detailed Heat and Fluid Flow Analysis of an Internal Permanent Magnet Synchronous Machine by Means of Computational Fluid Dynamics," *IEEE Trans. Ind. Electron.*, vol. 59, no. 12, pp. 4568–4578, 2012.

[15] Y. C. Chong, E. J. P. Echenique Subiabre, M. A. Mueller, J. Chick, D. A. Staton, and A. S. McDonald, "The Ventilation Effect on Stator Convective Heat Transfer of an Axial-Flux Permanent-Magnet Machine," *Industrial Electronics, IEEE Transactions on*, vol. 61, no. 8. pp. 4392–4403, 2014.

[16] P. H. Connor, S. J. Pickering, C. Gerada, C. N. Eastwick, and C. Micallef, "CFD modelling of an entire synchronous generator for improved thermal management," *Power Electronics, Machines and Drives (PEMD 2012), 6th IET International Conference on*. pp. 1–6, 2012.

[17] P. H. Connor, S. J. Pickering, C. Gerada, C. N. Eastwick, C. Micallef, and C. Tighe, "Computational fluid dynamics modelling of an entire synchronous generator for improved thermal management," *Electric Power Applications, IET*, vol. 7, no. 3. pp. 231–236, 2013.

[18] C. A. Cezario and A. A. M. Oliveira, "Electric motor internal fan system CFD validation," *Electrical Machines, 2008. ICEM 2008. 18th International Conference on*. pp. 1–6, 2008.

[19] C. A. Cezario and A. A. M. Oliveira, "CFD electric motor external fan system validation," *Electrical Machines, 2008. ICEM 2008. 18th International Conference on*. pp. 1–6, 2008.

[20] C. Cezário, "Análise do escoamento do ar em motores de indução totalmente fechados," 2007.

[21] J. M. Yu and R.-N. Guo, "Numerical Simulation of Whole Steady Flow Field of Centrifugal Fan," *Computational and Information Sciences (ICCIS), 2010 International Conference on*. pp. 246–249, 2010.

[22] Z. Jian-hui and Y. Chun-xin, "Design and Simulation of the CPU Fan and Heat Sinks," *Components and Packaging Technologies, IEEE Transactions on*, vol. 31, no. 4. pp. 890–903, 2008.

[23] Z. Jian-hui and Y. Chun-xin, "Parametric Design and Numerical Simulation of the Axial-Flow Fan for Electronic Devices," *Components and Packaging Technologies, IEEE Transactions on*, vol. 33, no. 2. pp. 287–298, 2010.

[24] Y. Yao, W. Zhang, B. Shi, and C. Wang, "Experiment and simulation research of engine cooling fan in dump truck," *Electron. Mech. Eng. Inf. Technol. (EMEIT), 2011 Int. Conf.*, vol. 2, pp. 742–745, 2011.

[25] B. Daly, *Woods practical guide to fan engineering*. 1978.

VI. Biographies

Unai SanAndres was born in Bilbao in December 1986. He received the B.S. degree in electrical engineering from the University of Mondragon, Mondragon, Spain, in 2010, where he is currently working toward the Ph.D. degree.

His current research interests include permanent magnet-machine thermal design and optimization.

Gaizka Almandoz was born in Arantza in March 1979. He received the B.S. and Ph.D. degrees in electrical engineering from the University of Mondragón, Mondragón, Spain, in 2003 and 2008, respectively.

Since 2003, he has been with the Department of Electronics, Faculty of Engineering, University of Mondragón, where he is currently an Associate Professor. His current research interests include electrical machine design, modeling, and control. He has participated in various research projects in the fields of wind energy systems, lift drives, and railway traction.

Javier Poza was born in Bergara, Spain, in June 1975. He received the B.S. degree in electrical engineering from the University of Mondragón, Mondragón, Spain, in 1999, and the Ph.D. degree in electrical engineering from the Institut National Polytechnique de Grenoble, Grenoble, France.

Since 2002, he has been with the Department of Electronics, Faculty of Engineering, University of Mondragón, where he is currently an Associate Professor. His current research interests include electrical machine design, modeling, and control. He has participated in various research projects in the fields of wind energy systems, lift drives, and railway traction.

Gaizka Ugalde received the B.Eng. and Ph.D. degrees in electrical engineering from the University of Mondragón, Mondragón, Spain, in 2006 and 2009, respectively.

Since 2009, he has been with the Department of Electronics, Faculty of Engineering, University of Mondragón, where he is currently an Associate Professor. His current research interests include permanent-magnet-machine design, modeling, and control. He has participated in various research projects in the fields of lift drives and railway traction.

Ana Julia Escalada was born in Pamplona, Spain, in April 1977. She received the B.S. degree in electronic engineering from the University of the Basque Country, Bilbao, Spain, in 2001, the B.Sc. degree in physics from the University of Cantabria, Santander, Spain, in 2003, and the Ph.D. degree in automatic and industrial electronic engineering from the University of Mondragón, Mondragón, Spain, in 2007, in conjunction with the Power Electronics Department, Ikerlan Technological Research Center.

She is currently with the Electrical Drives Department, ORONA Elevator Innovation Centre, Hernani, Spain. Her interests are drives and electrical machines for lifts.