25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

COB-2019-2049
COMPUTATIONAL FLUID DYNAMICS: COMPORTAMENTAL STUDY AND OPTIMIZATION OF A RADIAL FAN

Filipe dos Santos Aureliano
University Center of the Southern Minas, Unis Group; Joaquim Camilo Tavares street, 570
filipe254@hotmail.com

Luiz Carlos Vieira Guedes
University Center of the Southern Minas, Unis Group; Joaquim Camilo Tavares street, 570
guedes@unis.edu.br

Abstract. *Due to the absence of standards that specify the optimum sizing of industrial ventilators, companies end up scaling their products under strong influence from their experiences acquired over the years, that is, empirically it can cause oversizing of the ventilators, thus, incurring unnecessary expenses with low yields directly influencing the final product. The scope of this work is based on the computational fluid dynamic analysis of the radial ventilators, with qualitative and quantitative data on how these behave. The study was carried out from a physical prototype, using bench tests, In which characteristic data were collected to be inserted into the ANSYS FLUENT Software for simulated geometry, in which the preliminary result of the original geometry enabled the optimization of the project, making the final product more efficient, eliminating the overdimensioning and proving the feasibility of using the simulation in the industrial environment.*

Keywords: *Radial Fan, Computational Fluid Dynamics, Oversizing.*

1. INTRODUCTION

Ventilators are motor-flow machines that transfer energy to gases through the action of a rotor or propeller, they are mechanical devices used to convert mechanical energy applied to its axis, into fluid dynamic energy, described as potential energy of pressure and kinetics (Pantankar, 1980). Through the energy gained, the fluid (in the case of gas) becomes able to drain in ducts, overcoming the resistances that oppose its displacement, providing the desired flow rate as project.

This type of turbomachinery is widely used in products of the most varied applications, such as: temperature controllers, material conveyors through pipelines, as well as air distributors. In addition, they are present in the primary sector of industrial production, such as petrochemical, steel mills, mining, among many others (Ferziger, 2002).

The flow in centrifugal fans, as in any turbomachine, is one of the most complex existing in fluid dynamics. In most cases, this flow is entirely three-dimensional, with laminar/turbulent transition phenomena and detachment associated with the development of boundary layers. Complex mechanisms of viscous dissipation and vorticity generation are also present. The flows in the movable and fixed components of the ventilator interfere with each other causing non-permanent effects (Bhope et al., 2004).

However, companies end up scaling their products under a strong influence from their experiences acquired over the years, that is, empirically causing sub or oversizing, incurring unnecessary spending for both the manufacturer and the final consumer.

Consequently, market demand makes industries develop products with high quality and yield, then companies need to find alternatives that require less cost and project execution time. One of the tools used to solve these problems in recent decades is the fluid-dynamic numerical simulation, through computational fluid dynamics (CFD) software, obtaining qualitative and quantitative data from the project.

The purpose of this study is to optimize a radial fan of a particular company, from a real model tested on a bench with the aid of the ANSYS/FLUENT Software. As these are several variables involved, the study proposed to change the number of blades to achieve the correct flow of the Project, retaining both geometric characteristics of the rotor and original Volute, because it is not necessary to make changes to the design of the equipment in which this turbomachine is inserted.

2. MATERIALS AND METHODS

In this chapter, the methodology adopted for the study will be shown, according to the flowchart shown in figure 1 below.

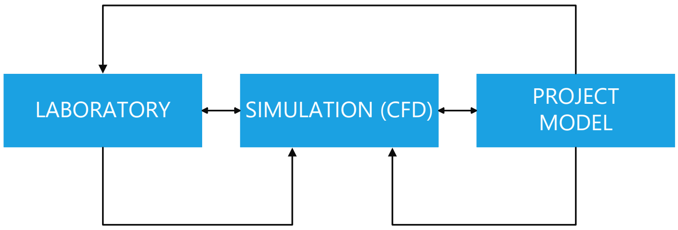

Figure 1. Flowchart of the research work methodology.
Available from: The authors.

The development of the experimental bench used to obtain the measures followed the following steps:

- Using the AC drive to control motor speed at 189,5 Rad/s;
- Use of anemometer at valve outlet to use fluid velocity;
- Pressure measurement with pressure gauge at volute outlet;

The measurements carried out of static pressure on the admission of the volute, which added to an estimation of the dynamic pressure (obtained from the fan flow), provided the total pressure on the intake of the fluid (air), the dynamic pressure measurements were also performed at the output of the volute, being the total pressure (because as the discharge is made in the atmosphere the relative static pressure is zero). Figure 2, schematically represents the procedure performed in the experimental trial.

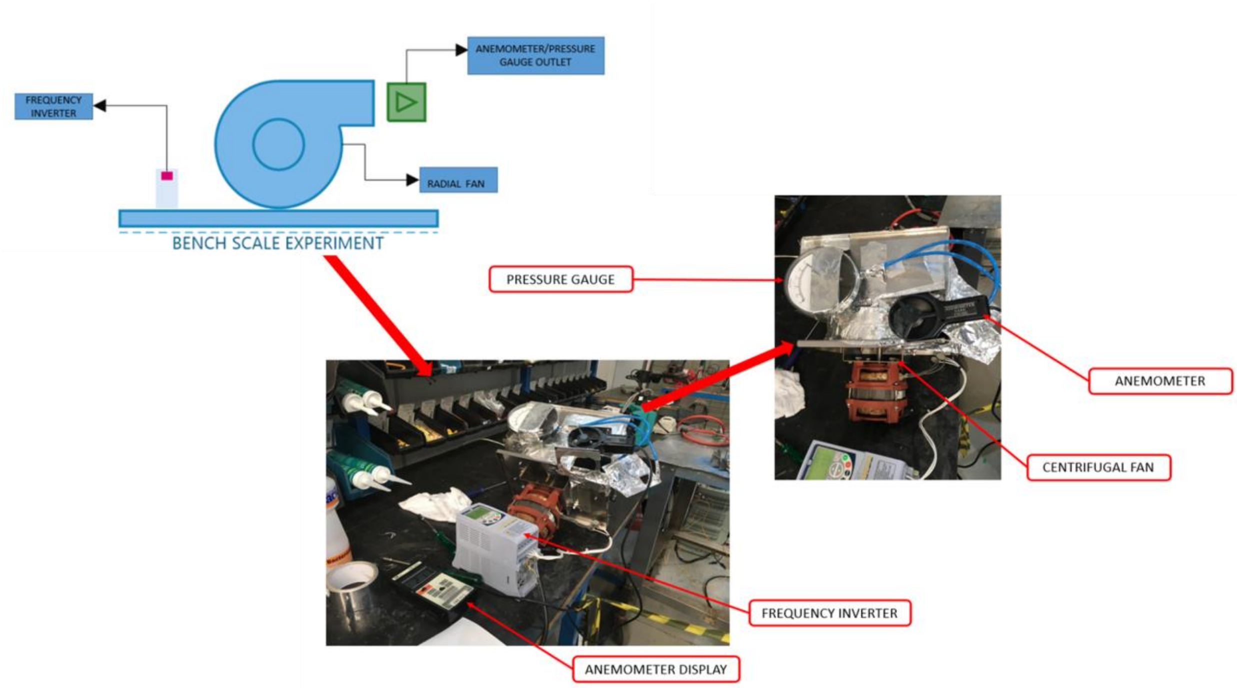

Figure 2. Experimental bench.
Available from: The authors.

For the solution of the case first was modeled the radial fan through a 3D CAD software, figure 3, following the measurements and the quantity of original blades of the actual model used on the bench.

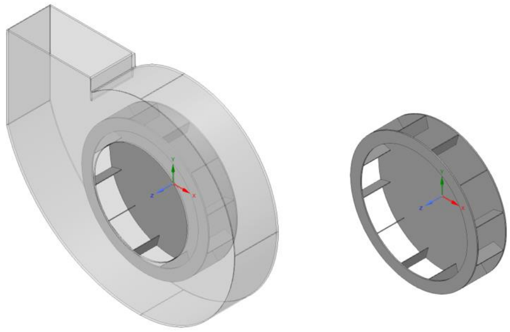

Figure 3. 3D modeling of radial fan.
Available from: The authors.

The domain used was divided into two regions, creating a domain for the snail, figure 4 (a), and the other for the rotor region, figure 4 (b).

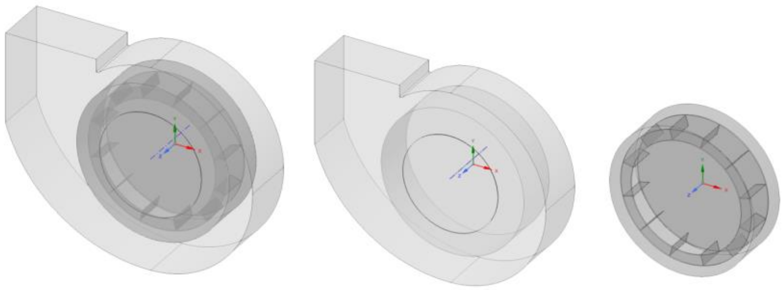

Figure 4. (a) domain for the volute; (b) domain for the rotor.
Available from: The authors

In this next step was generated the mesh, figure 5 below, with tetrahedral elements, prismatic surface layers near the walls.

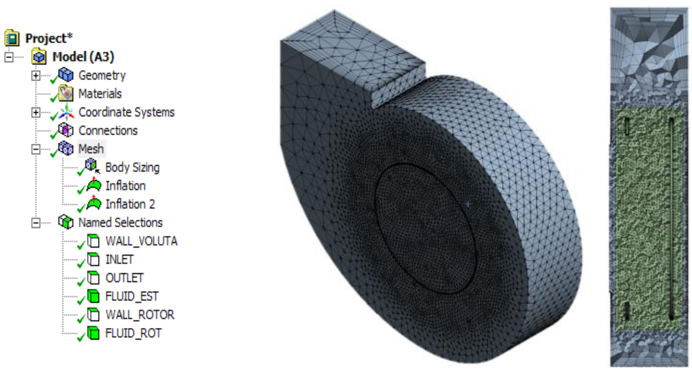

Figure 5. Fan mesh.
Available from: The authors.

To evaluate the quality of this discretization, we used the Skewness technique, figure 6, for being one of the main, determining how close to the ideal the mesh is, because a poor discretization in detail of the mesh can cause instability and errors in the solution.

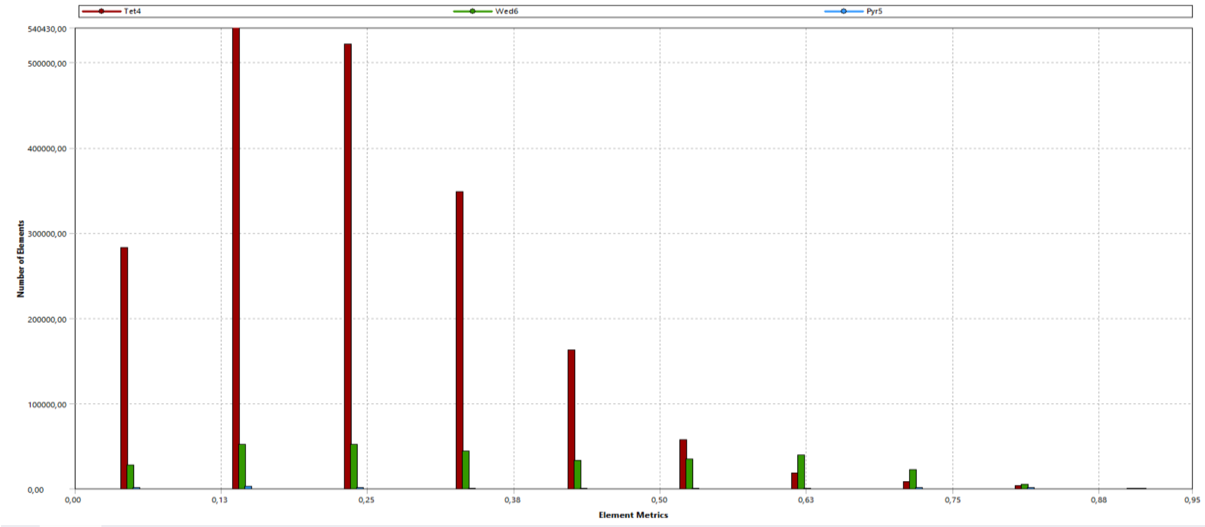

Figure 6. Skewness.
Available from: The authors.

The contour conditions adopted are atmospheric pressure at the inlet, atmospheric pressure at the output and rotation of 189.5 Rad/s in the rotating rotor domain. On the walls were adopted non-slip condition, so as to capture the viscous effects of the air, considering a 10% turbulence intensity and a permanent regime.

The method for numerical solution uses the integral form of the conservation equations as its starting point (Ferziger et al., 2002). Its working principle is given by the subdivision of the solution domain into a finite number of adjacent control volumes, where each control volume is applied the conservation equations, where then, an algebraic equation is obtained for each volume of control that is iteratively resolved for each volume (Patankar, 1980); (Maliska, 2004). The equations of mass conservation and amount of linear motion are resolved by means of the Navier-Stokes equations with Reynolds averages (RANS), as can be seen in (Wilcox, 1998). For the present work, we opted for the use of the turbulence model k-ω SST, a model that is a mixture between k-ε e k-ω models. According to Menter et al., (2003), the formulation is based on mixing functions, which ensures an adequate selection of k-ε e k-ω zones without user iteration.

3. RESULTS AND DISCUSSIONS

3.1 Numerical and experimental validation of the current model

In this stage, it was verified the truthfulness between the bench test and the simulated model, comparing the values obtained from pressure, speed and flow at the output of the volute. This comparison was used to validate the software setup, ensuring the reliability of the results obtained by these, as figure 7.

Methods	Blade Number	Rotor Velocity - Rad/s	Velocity OUTLET - m/s	Pressure Gradient OUTLET	MASS FLOW RATE OUTLET Kg/s
EXPERIMENTAL RESULTS	12 - ORIGINAL	185,9	15,31	189,121	0,18192
NUMERICAL RESULTS	12- ORIGINAL	185,9	16,44	192,077	0,18345

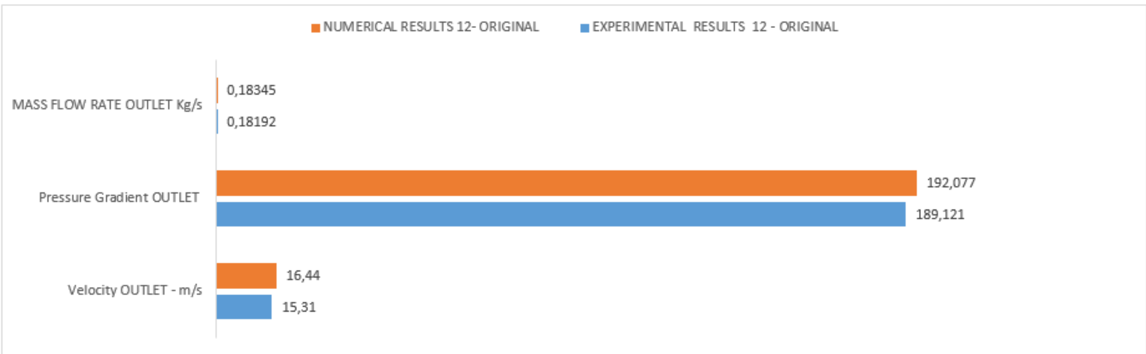

Figure 7. Experimental results and numerical simulation.
Available from: The authors.

According to the data obtained from the comparison between the real and the simulated model, detailed in the previous figure, the variation between the mass flow rate presented 0.834%, the output pressure was a variation of 1.539% and the output velocity presented 0.931%. The differences presented in the comparison of the numerical and experimental results were satisfactory, and the uncertainties of measurements can be up to 5% and, therefore, the differences are in the margin of error.

3.2 Numerical simulation of new models

The numerical simulations for the new models were performed ranging from 12 to 30 paddles following the original formats dnd for measuring efficiency, the flow rate was adopted. The results obtained can be seen in figure 8.

Blade Number	MASS FLOW RATE OUTLET Kg/s
12	0,18345
13	0,19036
14	0,19456
15	0,20188
16	0,20796
17	0,21295
18	0,21857
19	0,22156
20	0,22746
21	0,22998
22	0,23474
23	0,23897
24	0,24363
25	0,24549
26	0,24856
27	0,25113
28	0,25534
29	0,25754
30	0,25942

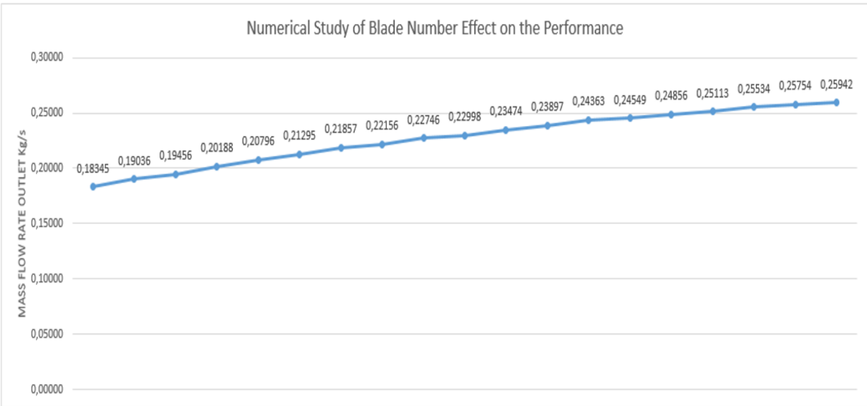

Figure 8. Numerical study of blade number.
Available from: The authors.

It is perceived that as the number of blades increases, the output flow increases. The increase of blades intuitively means to reduce the spacing between the blades, "narrowing" the path of the fluid and reducing centrifugal effects, thus improving fluid flow and increasing its yield.

Figure 9, below illustrates how pressure and speed behaves in the domain. The original design consisted of 12 paddles presenting a flow rate of 0.18345 kg/s, which did not satisfy the demand of the equipment. To meet the demand of 0.23 kg/s was adopted a rotor with 22 paddles that has a flow rate of 0.23474 kg/s.

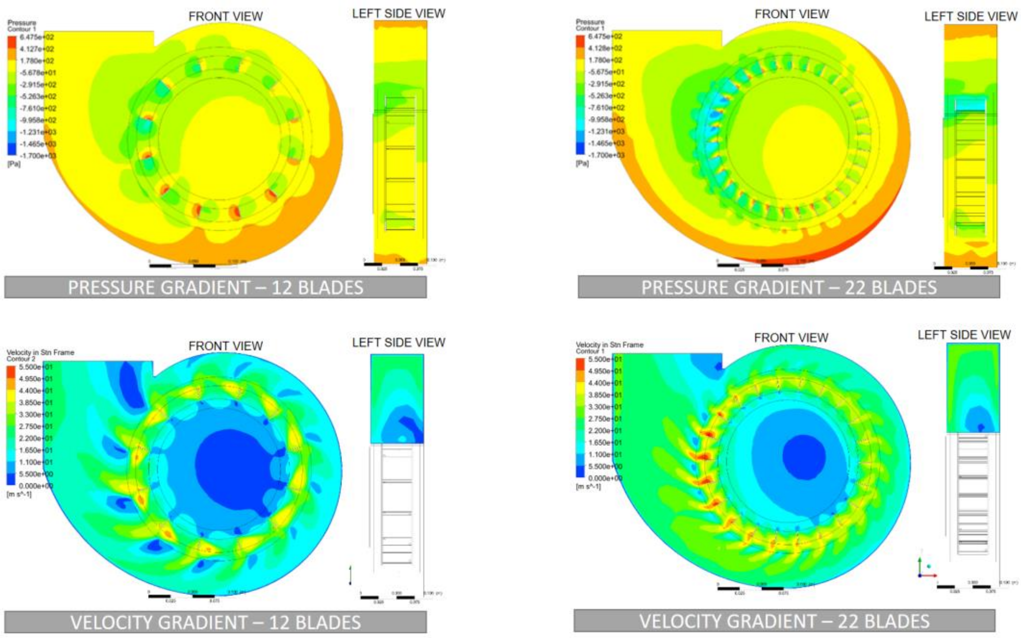

Figure 9. (a) pressure gradient; (b) velocity gradient.
Available from: The authors.

The aforementioned images show that due to the shape of the straight blades, they present regions of fluid recirculations and internal volumetric losses and, therefore, where further optimizations should be made to improve the performance of the equipment. These recirculations and vortices occurring inside the ventilator are responsible for a malfunction of the machine, high noise and loss of load

4. CONCLUSION

The present work presented a numerical and experimental analysis on the performance of a centrifugal blower of a particular company. The results obtained were satisfactory, being possible to obtain considerable improvements only in the modification of the number of blades of the same.

The great advantage of numerical application in the study was to obtain adequate flow without the need to change the original design of the equipment in which it is inserted, facilitating its implementation in the products already finished and in the manufacturing process.

In a comparative of the numerical and experimental results, there was a difference of 0.834% for Mass flow rate outlet, 1.539% for the output pressure and 0.931% output speed, presenting an error less than 5% adopted, relatively small for the same.

It is noteworthy that in addition to the characteristic curves, the numerical approach allowed to check the field of velocity and pressure inside the ventilator, Information that is not found with the experimental approach, mainly due to the difficulty of instrumenting the inside of the rotor with pressure and velocity sensors.

5. REFERENCES

Bhope, D. V.; Padole P. M., 2004. “Experimental and theoretical analysis of stresses, noise and flow in centrifugal fan impeller, Mechanism and Machine Theory.”. Vol. 01, pp. 1257-1271.

Ferziger, J. H.; Peric, M, 2002. “Computational Methods for Fluid Dynamics.” Springer, Berlin, 3th edition.

Maliska, C. R., 2004. “Transferência de Calor e Mecânica dos Fluidos Computacional.” .Rio de Janeiro, 2nd edition.

Menter, F. R.; Kuntz, M.; Langtry, R., 2003. “Ten Years of Industrial Experience with the SST Turbulence Model, Turbulence Heat and Mass Transfer.”

Patankar, S. V., 1980. “Numerical Heat Transfer and Fluid Flows.” McGraw-Hill, New York, USA.

Wilcox, D. C., 1998. “Turbulence Modeling for CFD.” DCW Industries, Inc.

6. RESPONSIBILITY NOTICE