17th Brazilian Congress of Thermal Sciences and Engineering
November 25th-28th, 2018, Águas de Lindóia, SP, Brazil

ENCIT-2018-0007
NUMERICAL AND EXPERIMENTAL ANALYSIS OF RADIAL FANS APPLIED IN AGRICULTURAL SPREADERS USING CFD

Marcelo Luiz de Freitas Fogal
Gustavo Barbosa Micheli
Depto. de Engenharia Mecânica - Universidade Estadual Paulista Julio de Mesquita Filho - FE/UNESP, Bauru, SP, Brasil
marcelo.fogal@hotmail.com
gustavobm.micheli@gmail.com

Vicente Luiz Scalon
Depto. de Engenharia Mecânica - Universidade Estadual Paulista Julio de Mesquita Filho - FE/UNESP, Bauru, SP, Brasil
scalon@feb.unesp.br

Alcides Padilha
Depto. de Engenharia Mecânica - Universidade Estadual Paulista Julio de Mesquita Filho - FE/UNESP, Bauru, SP, Brasil
padilha@feb.unesp.br

Abstract. *This work presents a comparison between the numerical and experimental results of the analyses of two different types of radial rotors used in an agricultural fertilization system at a rotation equal to 4000 rpm. To validate the numerical analysis, experiments were carried out on a test bench using a hot-wire anemometer for velocity measurements and a Pitot tube for the pressure measurements according to the standard for laboratory tests. The turbulent mean field was obtained with the Reynolds Averaged Navier Stokes (RANS) and the turbulence model used to close the set of equations was the k-ε model of two equations. The computational fluid dynamics code CFX 18.1 was used to obtain the solutions of the transport equations, which uses the Finite Volume technique as a numerical method. In all models, we used non-structured tetrahedral meshes generated by the commercial software ICEM CFD 18.1. The main results showed that the methodology employed is adequate and able to reproduce the fluid dynamics behavior of the air flow.*

Keywords: *Radial fan, computational fluid dynamics, turbulence model, hot-wire anemometer, Pitot tube.*

1. INTRODUCTION

Agriculture is an essential activity for world food production and due to the large population increase, farmers need to produce more and in smaller area. One way to increase production is to use high-yield agricultural equipment (Camargo, 2012). Some of these equipments can be used for agricultural fertilization and many of these machines use the airflow to transport the particulate (fertilizer) to the application outlets providing a uniform and precise distribution. For the generation of this flow, machines need a fan coupled to the system which is used as vehicle to create the energetic gradient that allows the desired air flow.

To improve the quality of application, companies invest in studies of fertilizing systems optimization. These studies may be directed at increasing fan efficiency, which provides a better entire system efficiency. In order to determine the efficiency of a fan or a given system, experimental tests or a mathematical modeling can be done using some numerical method. There are several numerical methods applicable to the simulation of flows, among which the most known is the Finite Volume method, which is the method used in the present work.

The use of computational fluid dynamics (CFD) allows engineers to obtain numerical solutions to problems with complex geometries and imposed boundary conditions. In the literature there are several works on the subject such as Kergourlay et al (2006), which based on the numerical simulations of Kouidri et al (2005) performed an experimental study applied to axial fans, and as Yang et al. (2007) investigation with radial fans.

A CFD analysis can determine the values of velocity, temperature and pressure over a given domain. The rotor simulation can be done by dividing the domain into two: one static, represented by the volute, and the other rotating, representing the rotor. Kelecy (2000) describes the use of rotational subdomains connected with stationary domains resulting in good approximations for the flow promoted by the rotor, presenting very good correlation with experimental data. A major advantage of the CFD technique is its flexibility to change process parameters, flow regimes and scroll geometry, without performing a full-scale prototype.

In this way the work proposes a comparison between the numerical and experimental results of the analyses of two different types of radial rotors (A1 and B1) used in an agricultural fertilization system.

2. METHODOLOGY

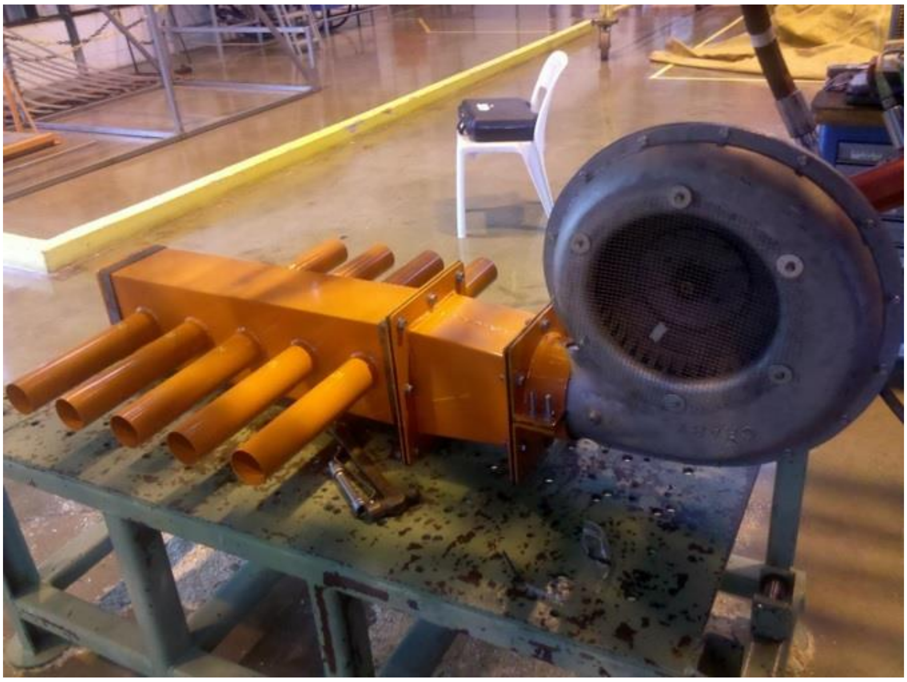
The mathematical model for the fluid used in this work is based on the Navier-Stokes equations, being considered three-dimensional flow, permanent regime, compressible fluid with ideal gas behavior, turbulent flow, and neglected the influence of the gravitational field. The turbulent mean field was obtained with the Reynolds Averaged Navier Stokes (RANS) and the turbulence model used to close the set of equations was the k-ε model of two equations. In order to obtain the solutions of the transport equations, the computational fluid dynamics code CFX 18.1 was used, which uses the Finite Volume technique as numerical method. In all models, we used non-structured tetrahedral meshes generated by the commercial software ICEM CFD 18.1.

To validate the numerical results, experiments were carried out on a test bench (Figure 1 and 2) using a hot-wire anemometer for velocity measurements and a Pitot tube for the pressure measurements according to the standard for laboratory tests (ANSI / AMCA 210-99). The velocity, pressure and temperature measurements were performed at the outlets and the values of torque and fan rotation were monitored to obtain the power consumed. The system efficiency was computed from these parameters.

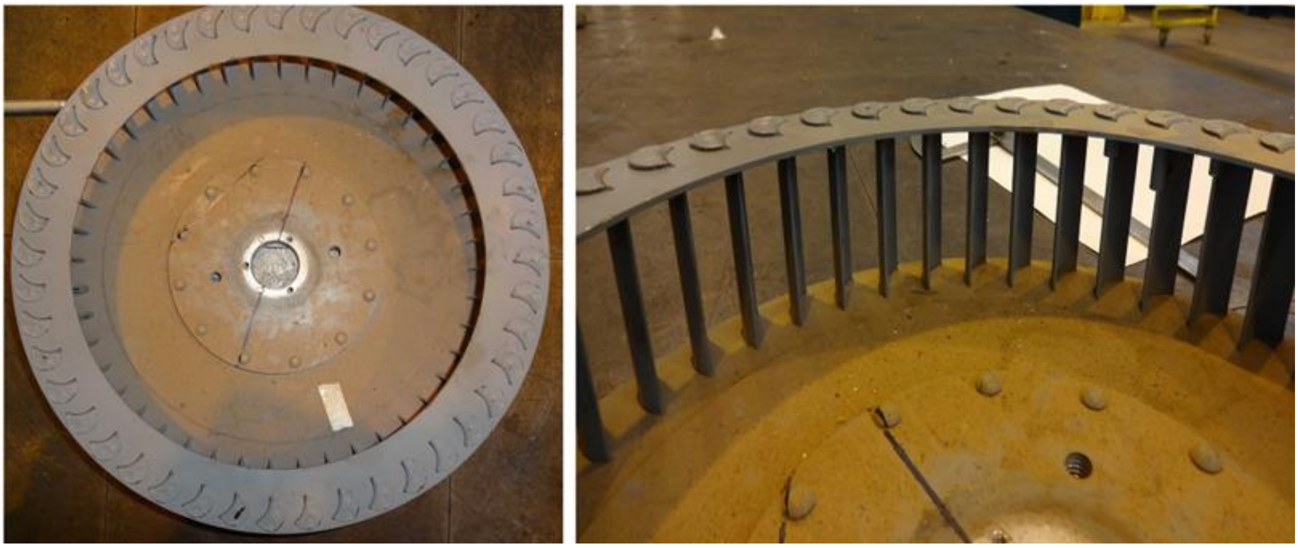
(a)

(b)

Figure 1. (a) test bench with Rotor A1; (b) Rotor A1 detail.



(a)



(b)

Figure 2. (a) test bench with Rotor B1; (b) Rotor B1 detail.

3. MATHEMATICAL MODEL

The finite volume method is extensively described in the literature in works such as Moreau & Bennett (1997), Hill & Wyman (1997) and Maliska (2004) and its application is discussed in recent works such as Vibhakar (2012). The authors discuss topics such as numerical discretization and consequent obtaining linearized equations and methods for solving the resulting equations.

3.1. Transport Equations

The governing equations are integrated over each control volume, such that the relevant quantity (mass, movement, energy, etc.) is conserved for each control volume. The equations solved for the discretized flow domain are the mass conservation (Eq. (1)), motion (Eq. (2)) and energy (Eq. (3)):

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}(\rho \overline{U_j})=0 \qquad (1)$$

$$\frac{\partial}{\partial t}\left(\rho\overline{U_i}\right)+\frac{\partial}{\partial x_j}\left(\rho\overline{U_i U_j}\right)=-\frac{\partial\overline{P^*}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{ef}\left(\frac{\partial\overline{U_i}}{\partial x_j}+\frac{\partial\overline{U_j}}{\partial x_i}\right)\right]+\rho f_i \tag{2}$$

$$\frac{\partial}{\partial t}(\rho\overline{H})-\frac{\partial P}{\partial t}+\frac{\partial}{\partial x_j}(\rho\overline{U_j}\,\overline{H})=\frac{\partial}{\partial x_j}\left(\lambda\frac{\partial\overline{T}}{\partial x_j}+\frac{\mu_t}{\mathrm{Pr}_t}\frac{\partial h}{\partial x_j}\right)+\rho\overline{U_i}f_i+\frac{\partial}{\partial x_j}\left\{\overline{U_i}\left[\mu_{ef}\left(\frac{\partial\overline{U_i}}{\partial x_j}+\frac{\partial\overline{U_j}}{\partial x_i}\right)-\frac{2}{3}\rho\delta_{ij}k\right]+\mu\frac{\partial k}{\partial x_j}\right\} \tag{3}$$

where ρ is the fluid specific mass, t is the time, U_i and U_j are the velocity vectors, x_i and x_j are the spatial components in Cartesian coordinates, P is the pressure, μ_{ef} is the effective fluid viscosity, f_i is related to field forces, Pr_t is the turbulent Prandtl number, k is the turbulent kinetic energy, λ is the thermal conductivity, H is the total enthalpy, h is the static enthalpy, T is the temperature and δ_{ij} is the Kronecker delta.

In the k-ε model the transport equations used to obtain the local values of k and ε are respectively Equations 4 and 5.

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial\left(\rho\overline{U}_j k\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\Gamma_k\frac{\partial k}{\partial x_j}\right)+P_k-\rho\varepsilon \tag{4}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial\left(\rho\overline{U}_j\varepsilon\right)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\Gamma_E\frac{\partial\varepsilon}{\partial x_j}\right)+\frac{\varepsilon}{k}(C_{E_1}P_k-\rho C_{E_2}\varepsilon) \tag{5}$$

where Γ_k and Γ_E are the diffusive coefficients, P_k is the rate of production of the turbulent kinetic energy and C_{E1} and C_{E2} are the constants of the model.

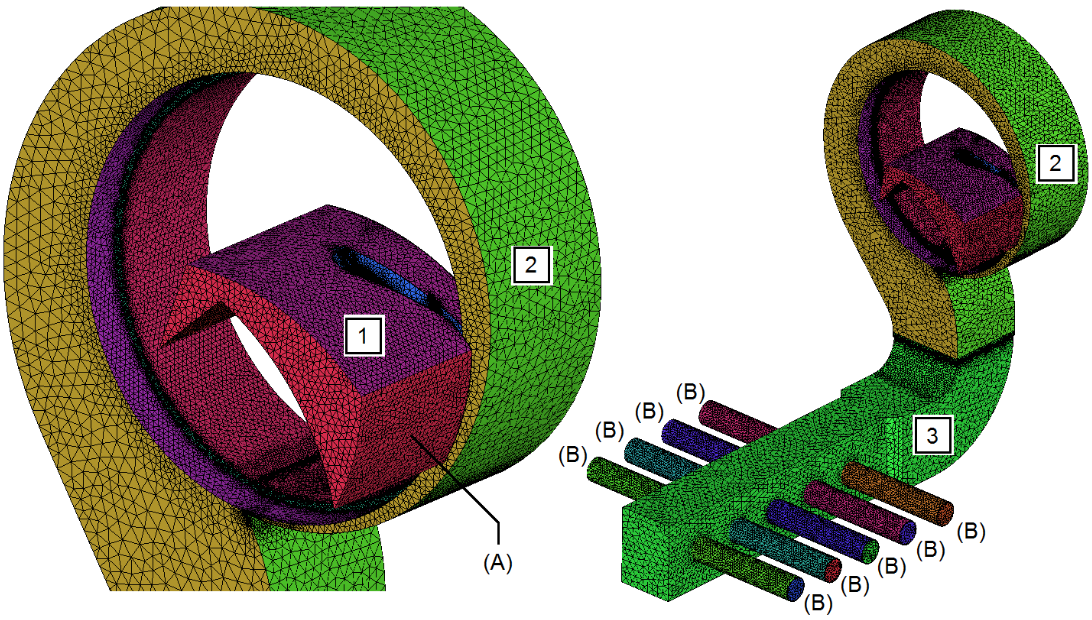
3.2. Mesh and Boundary Conditions

Due to the complexity of the geometries in the rotating domain, the software ANSYS ICEM 18.1 generated the meshes because it allows more control in the mesh creation.

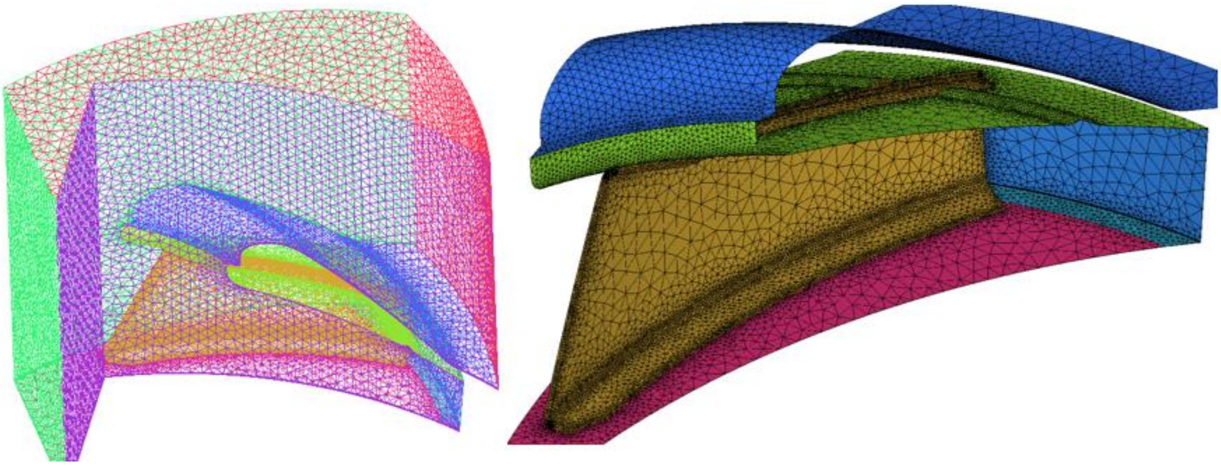
The mesh used in all models was a non-structured tetrahedral mesh, and it is easier to refine it in regions of geometric complexity. Contreras (2002) and Yang & Jiang (2010) detail the study of structured, unstructured and semi-structured meshes. A mesh refinement with layers of prisms was done on the wall to ensure adequate representation of the physical problem in the boundary layer (Joaquim-Jr et al., 2011). Souza (2011) presents a study of mesh refinement in a turbomachinery computational model.

The time step option used was Local Timescale Factor, which allows a different time step for each control volume until it reaches the pre-established convergence criterion (residue between iterations less than 10^{-4}).

Each model is composed of three subdomains: a rotating subdomain representing the rotor (1) and two downstream stationary subdomains representing the volute (2) and the air distribution box (3) as shown in figures 3 and 4.

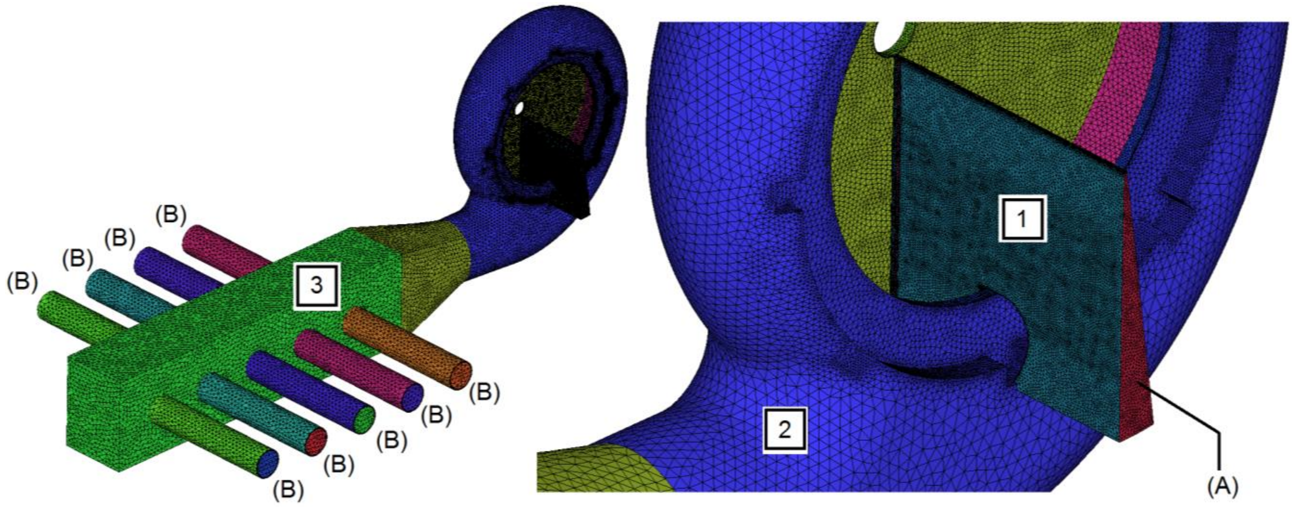


(a)

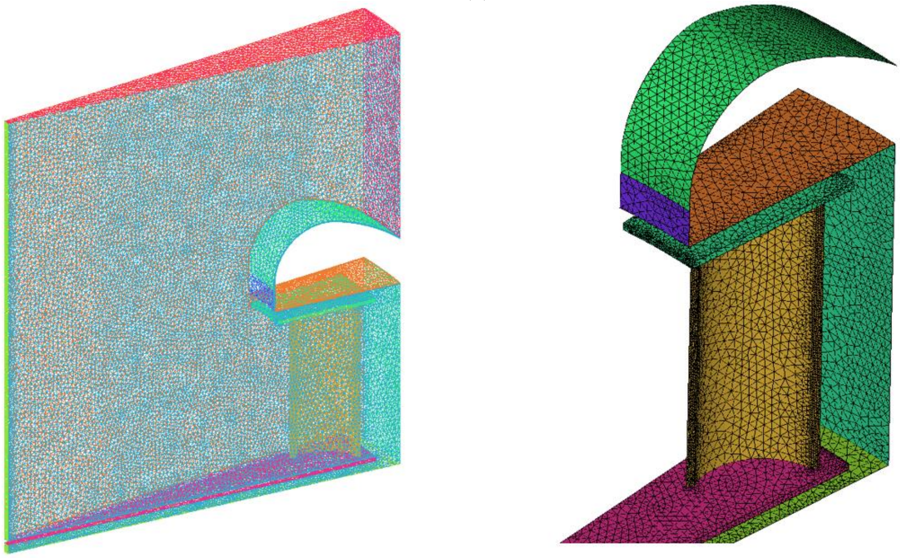


(b)

Figure 3. (a) A1 model; (b) rotating domain detail.



(a)



(b)

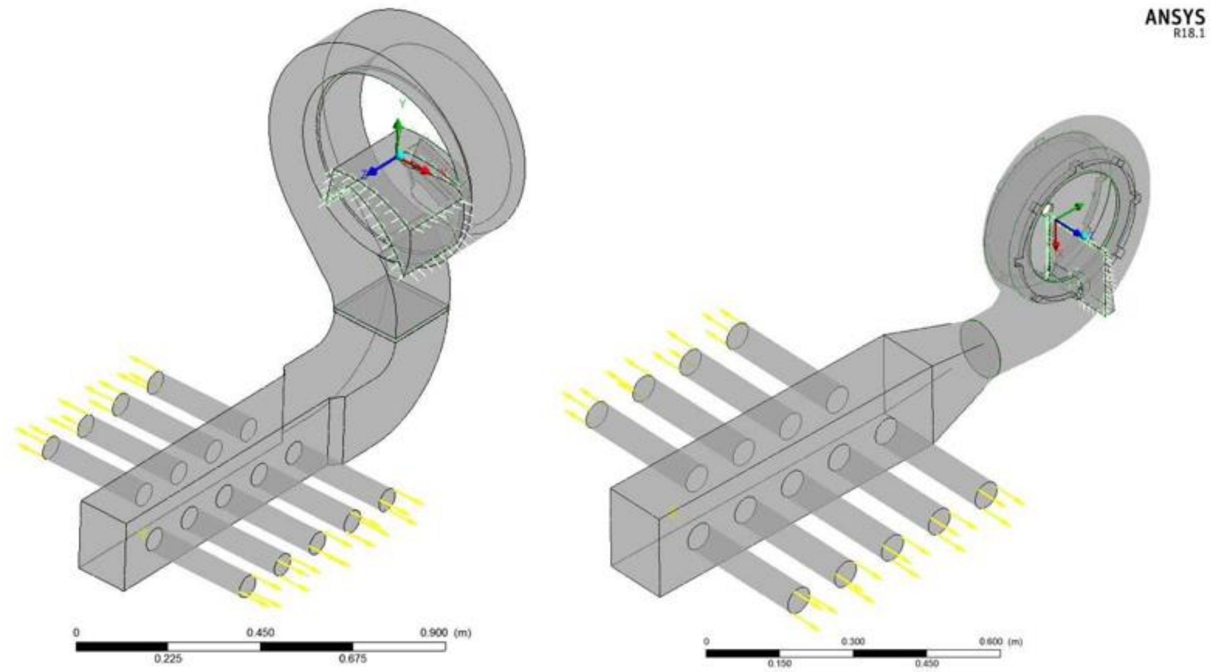
Figure 4. (a) B1 model; (b) rotating domain detail.

To reduce the number of elements and processing time used a periodicity interface to represent the rotating domain reduced to a blade-occupied angle. The interface between subdomains 1-2 is fluid-fluid interfaces with angle variation and the interface between subdomains 2-3 is fluid-fluid interfaces with automatic connection.

Boundary conditions:

• Inlet A: relative total pressure zero and temperature 26.2ºC (experimental temperature);
• Outlet B: relative static pressure zero;
• Stationary domains walls: zero velocity and no slip/smooth wall condition;
• Rotating domains walls: speed of 4000 rpm, no slip/smooth wall condition.

Figure 5 shows the detail of the contour conditions.

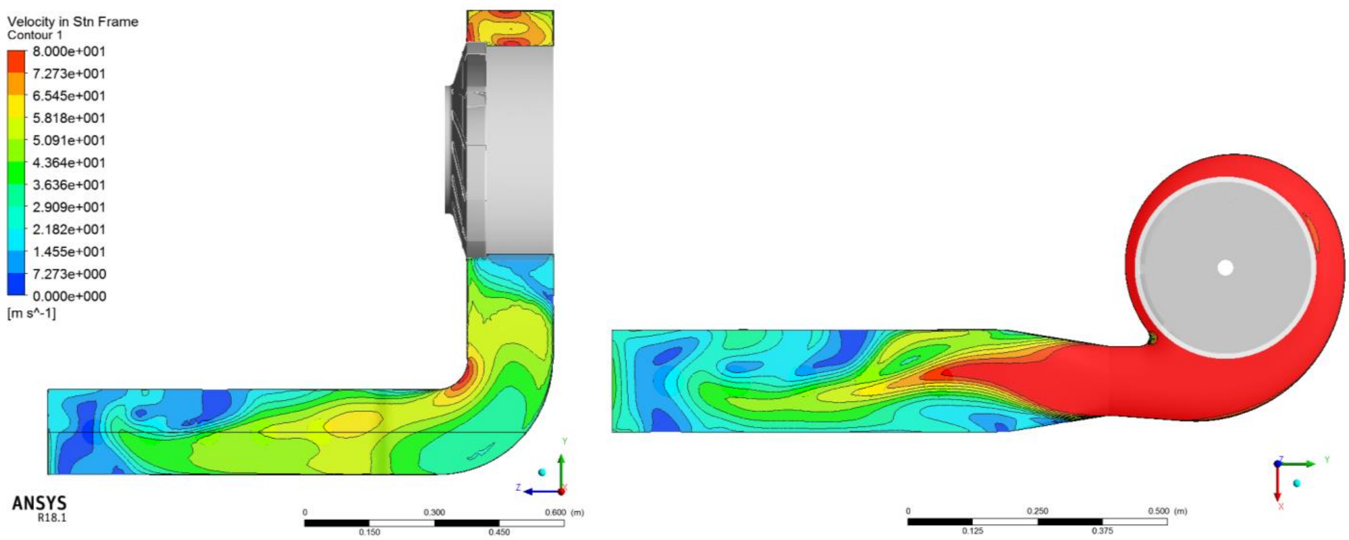


(a) (b)

Figure 5. (a) A1 boundary conditions; (b) B1 boundary conditions.

4. RESULTS AND DISCUSSIONS

The figures below show the comparison between the air distribution systems in A1 and B1 models.



(a) (b)

Figure 6. Velocity gradient in the central plane with lateral view; (a) A1; (b) B1.

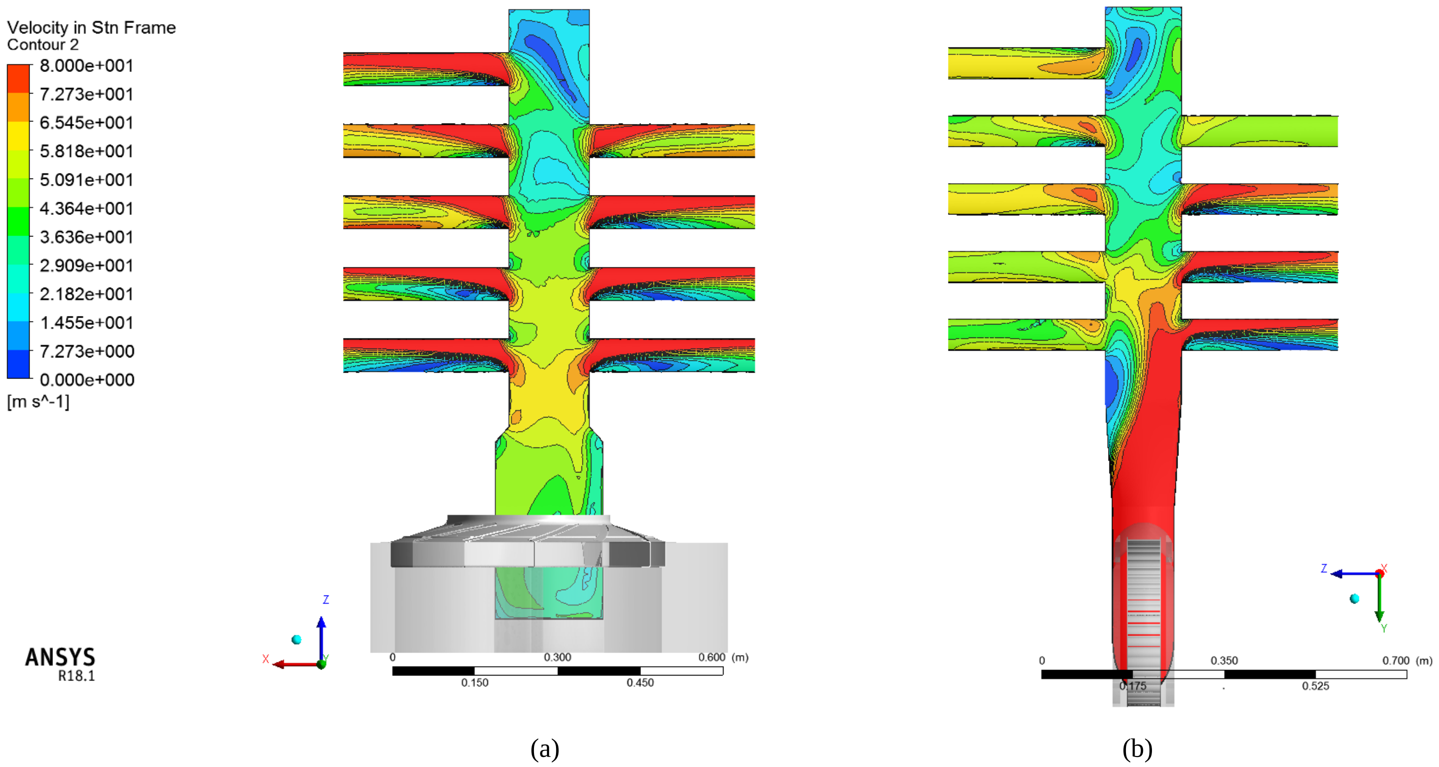


Figure 7. Velocity gradient in the central plane with top view; (a) A1; (b) B1.

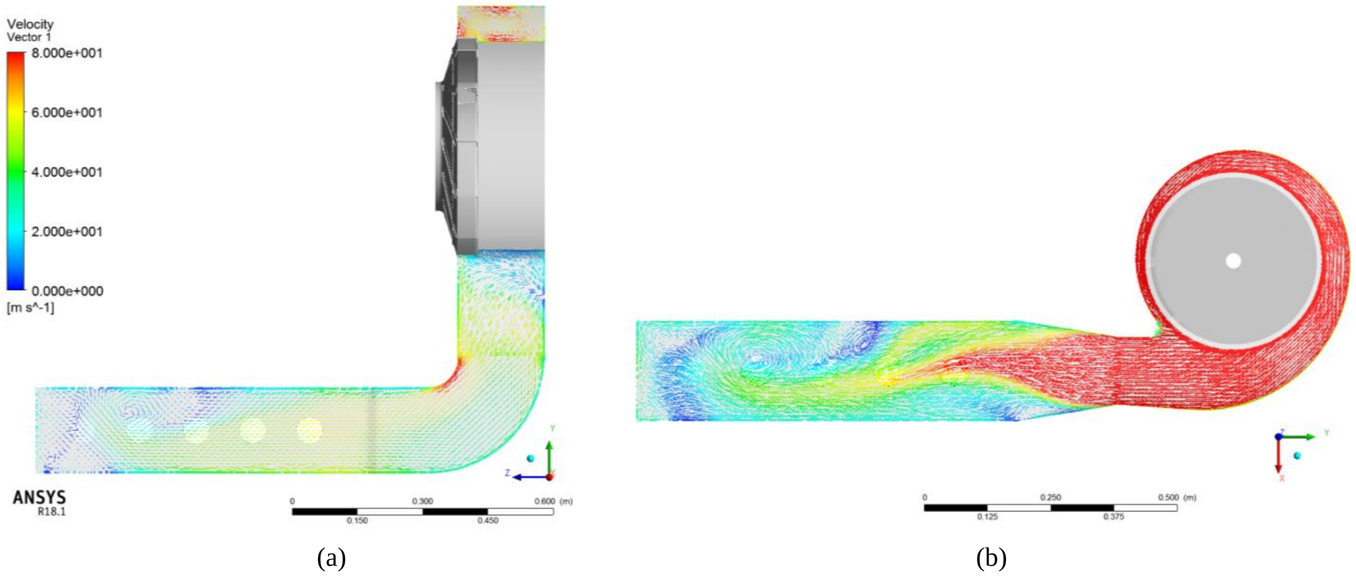
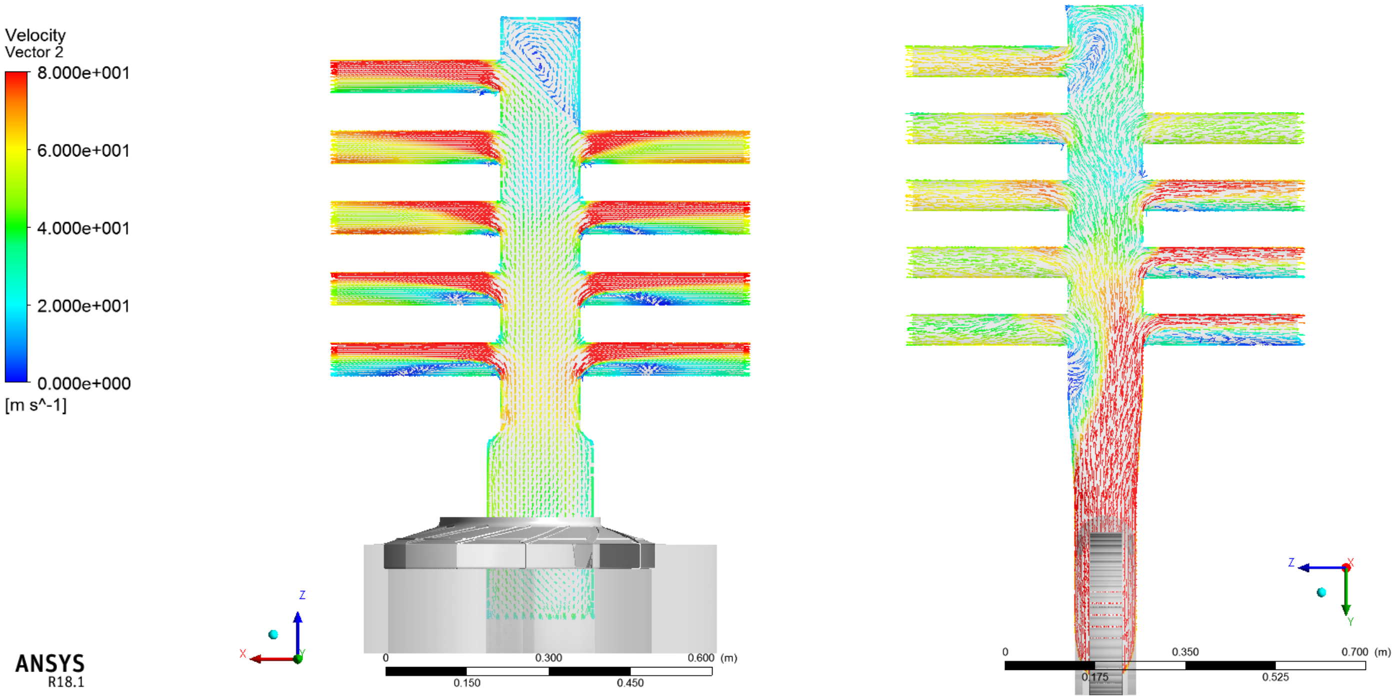


Figure 8. Velocity vectors in the central plane with lateral view; (a) A1; (b) B1.

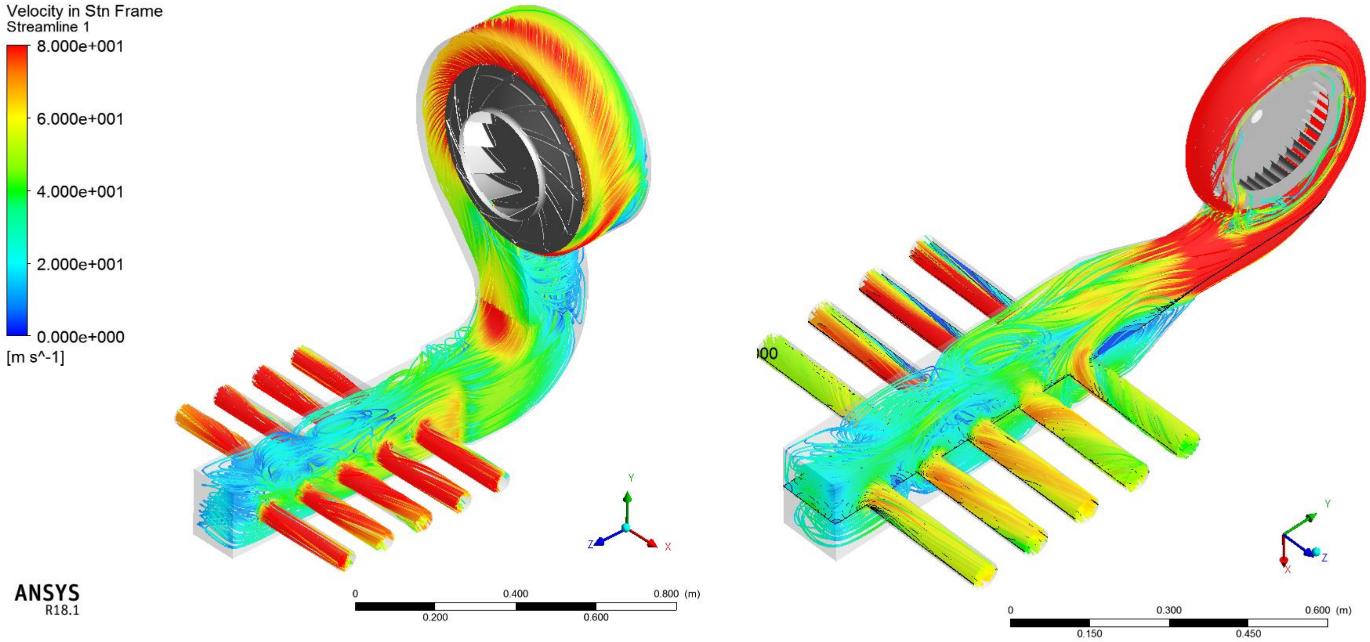
The numerical results show from figures 6 to 10 that the Rotor A1 presents fewer regions with recirculations and a uniform distribution of air in the outlets in relation to the Rotor B1. Thus, as shown in table 1, the Rotor A1 shows a greater efficiency in relation to the Rotor B1 for this system at rotation of 4000 rpm.

The velocity values in the volute of Rotor B1 are higher than the values of Rotor A1. Perhaps in another application with lower rotation (reducing the loss of load) and an air distribution box with the similar diameter of the volute results in a higher efficiency for this system with Rotor B1.



(a) (b)

Figure 9. Velocity vectors in the central plane with top view; (a) A1; (b) B1.



(a) (b)

Figure 10. Streamlines; (a) A1; (b) B1.

The comparison of the parameters between numerical and experimental results are in the same conditions and the table below presents the results.

The parameter that presented the greatest difference between the numerical and experimental results in the Rotor A1 was the power consumed (9.2%) and the Rotor B1 was the efficiency (5.2%). This difference between the variables may be related to mesh refinement, convergence criterion (residue between iterations less than 10^{-4}), turbulence model adopted or experimental uncertainties.

Table 1. Comparison between numerical and experimental results.

	Rotor A1			Rotor B1		
	Experimental	Numerical	Difference [%]	Experimental	Numerical	Difference[%]
Efficiency [%]	31,2	31,1	0,3	19,1	18,2	5,2
Flow rate [10^{-1}.m^3/s]	17,3	16,0	8,1	13,2	12,8	3,4
Power In [kW]	13,9	12,7	9,2	11,6	11,7	0,5
Outlet temperature [C]	34,7	33,4	3,8	35,2	34,4	2,2
Pressure [10^2.Pa]	25,1	24,7	1,3	16,7	16,5	1,1

The average difference between A1 and B1 rotors variables was around 4.5% and 2.5% respectively. This shows that the results are adequate for this type of analysis because a mesh refinement or a change in the convergence criterion and turbulence model would increase the computational cost.

Figure 11 shows the condensed table data.

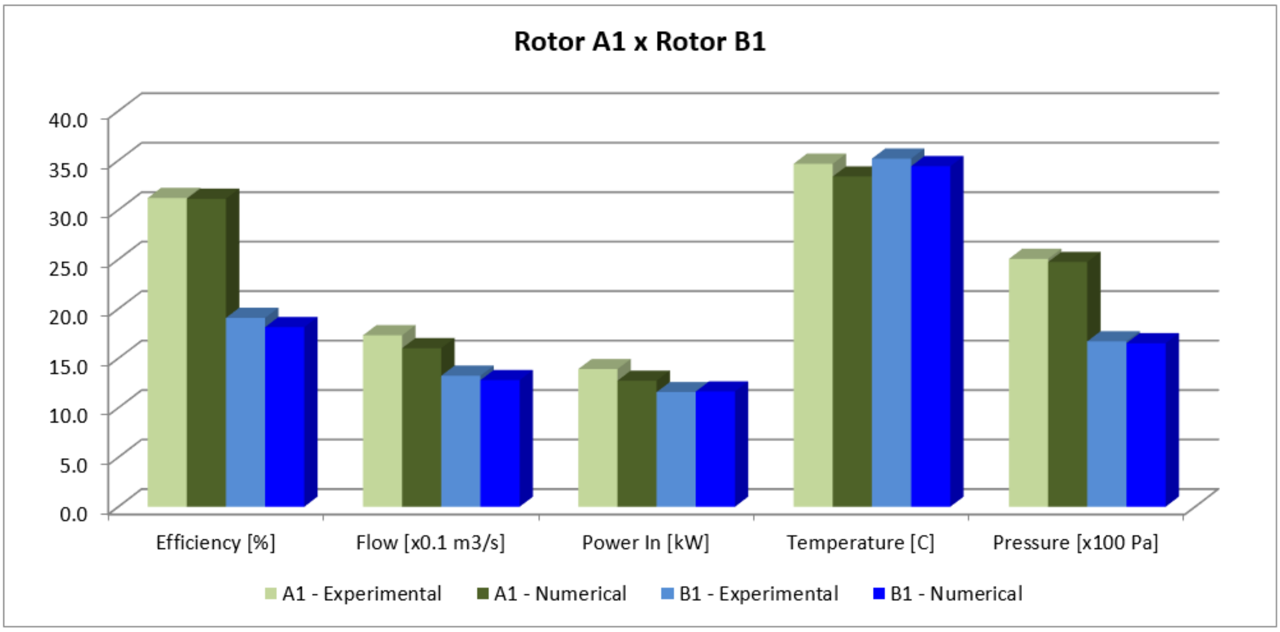


Figure 11. Numerical and experimental results for analyzed variables at rotation of 4000 rpm.

5. CONCLUSIONS

The results showed that the average differences between the evaluated parameters of flow, pressure, power consumed, temperature and system efficiency comparing the numerical and experimental methods was around 4.5% for one type of rotor and 2.5% to another. The CFX software presented very reliable results, close to those measured experimentally. In this way it can be considered that the methodology employed is adequate and able to reproduce the fluid dynamics behavior of the air flow for this type of application and can be replicated in other types of rotors to evaluate its operational efficiency without producing new full-scale prototypes. It was demonstrated the usefulness of the CFD methods for the advanced engineering project, and for this work, the k-ε model was very adequate for the survey of the general flow profile.

6. REFERENCES

ANSI/AMCA 210-99, An American National Standard, "Laboratory Methods of Testing Fans for Aerodynamic Performance Rating", 1999.

Camargo, M. S., “A importância do uso de fertilizantes para o meio ambiente” *Pesquisa & Tecnologia, Vol. 9*, n. 2, Jul-Dez, 2012.

Contreras, J., “Semi-unstructured grid methods for turbomachinery applications” ASME TURBO EXPO, Amsterdam, (GT2002-30572), 2002.

Hill, D. L. & Wyman, N., “A realistic prediction of the axial fan problem”, ASME Fluids Engineering Division Summer Meeting, USA, 1997.

Kelecy, F., “Study demonstrates that simulation can accurately predict fan performance”, *Journal Articles by Fluent Software Users,* New Hampshire- USA, 2000.

Kergourlay, G., Kouidri, S., Rankin, G. W. & Rey, R., “Experimental investigation of the 3D unsteady flow field downstream of axial fans*”, Flow Measurement and Instrumentation,* 17 pp. 303-314, 2006.

Kouidri, S., Fedala, D., Belamri, T. & Rey, R., “Comparative study of the aeroacoustic behavior of three axial flow fans with different sweeps”. In: FEDSM 2005. June 2005.

Joaquim-Jr., C. F., Reynol, A., Cekinski, E., Seckler, M. M., Nunhez, J. R., “Development of static mixers for miscible fluids in laminar flow with the use of computational fluid dynamics”, The Canadian Journal of Chemical Engineering, v89, p.734-744, 2011.

Maliska, C. R., “Transferência de calor e Mecânica dos Fluidos Computacional”, *2ª edição. Livros Técnicos e Científicos,* Editora S. A., Rio de Janeiro, 2004.

Moreau, S. & Bennett, E., “Improvement of fan design using CFD”, Society of Automotive Engineers, USA, 1997.

Vibhakar, N. N., “Studies on radial tipped centrifugal fan” *Thesis of Doctor of Science,* Veer Narmad South Ghjarat University, Surat, India, 350p, 2012.

Yang, H. & Jiang, L., “A dual mesh approach to enhance accuracy of the boundary conditions for unstructured grid modelling of turbomachinery flows” ASME TURBO EXPO, Glasgow, (GT2010-23390), 2010.

Yang, L., Hua, O. & Zhao-hui, D., “Experimental research on aerodynamic performance and exit flow field of low pressure axial flow fan with circumferential skewed blades” *Journal of Hydrodynamics,* Shanghai Jiaotong University, China, pp. 579-586, 2007.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.