THEORETICAL AND EXPERIMENTAL STUDY OF THE CHARACTERISTIC CURVES OF AN AXIAL FAN APPLIED TO AGRICULTURAL SPRAYING

Marcelo Luiz Freitas Fogal, marcelo.fogal@hotmail.com
Alcides Padilha, padilha@feb.unesp.br
Depto. de Engenharia Mecânica - Universidade Estadual Paulista Julio de Mesquita Filho - FE/UNESP, Bauru, SP, Brasil

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

Abstract. *In agricultural spraying, many machines need the fan to establish the flow of air that contributes towards: a reduction of drift and losses to the soil, an increase in deposits and coverage of the lower layer of leaves, improved penetration of spraying drops as a possibility to reduce doses and application volumes. In order to improve the application quality of agrochemicals, companies invest in spraying system optimization studies. These studies can be directed towards an increase in fan efficiency, which provides improved performance for the entire system. In order to determine the efficiency of a fan or a specific spraying system, experimental assays or mathematical modeling can be run using some sort of numerical method. This paper presents a theoretical and experimental analysis of comparative results between the characteristic curves of an axial fan used in an agricultural spraying system for a blade attack angle of 32 degrees at rotations of 1500, 1750, 2600 and 3000 rpm and numerical results for the influence of blade attack angle variation at 28, 32 and 36 degrees and optimization of the spraying system, both for a rotation of 2600 rpm. The mathematical model used in this study is based on Navier-Stokes equations, with mass balance always adjusted by the continuity equation. Flow was considered three-dimensional, turbulent and isothermal in a steady state, disregarding the influence of the gravity field. The fluid was considered viscous and non-compressible. This supposition of noncompressible fluid is needed since flow pressure variations do not imply significant variations in air density in operating conditions. The average turbulent field was obtained from the application of time average where the turbulence model required for closing the set of equations was the k-E model for two equations. Resolution of all connected phenomena was achieved with the help of the fluid dynamics computer, CFX, which uses the finite volumes technique as a numerical method. In order to validate the theoretical analysis, an experiment was conducted in a circular section of a horizontal wind tunnel, 622 mm in diameter and 6220 mm in length, using a Pitot tube for pressure readings according to the norm for laboratory assays. Qualitative results are shown as vectors and gradient maps for speed and quantitative results are shown in tables and graphs for characteristic curves. The main results demonstrate that the methodology used, based on CFD (Computational Fluid Dynamics) techniques, is able to reproduce the phenomenological behavior of an axial fan in a spraying system because the results were very reliable and similar to experimentally measured ones. Using mesh choice parameters and appropriate turbulence models and advection schemes, it is possible to obtain models that can numerically predict the behavior of a flow machine, becoming a very important tool in the project. For this study, the k-E model was sufficiently satisfactory for finding the general behavior flow.*

Keywords: *Axial fan, characteristic curves, Computational Fluid Dynamics, turbulence model, wind tunnel.*

1. INTRODUCTION

A lot of machines used in agricultural spraying need a fan as a vehicle to create the energy gradient that allows the desired flow of air. Assisting the air contributes towards: a reduction of drift and losses to the soil, an increase in deposits and coverage of the lower layer of leaves, improved penetration of spraying drops as a possibility to reduce doses and application volumes. In order to improve the application quality of agrochemicals, companies invest in spraying system optimization studies. These studies can be directed towards an increase in fan efficiency, which provides improved performance for the entire system.

An experimental test or mathematical modeling using some numeric method can be run to determine the efficiency of a fan or of a certain spraying system. In this study the finite volume method is used. The numeric simulation of flow machines can be done separating the domain in two: a stationary one, represented by the spraying system, and a rotational one, represented by the rotor. As described by Kelecy (2000), the use of rotational subdomains connected to stationary domains results in good approaches for the flow promoted by the fan, presenting very good correlation with experimental data.

For turbulence problems like this one, it is necessary to find ways to modify this equation that take this phenomenon into consideration. Although turbulence is a transient and chaotic phenomenon, it is possible to assess its permanent average effects by adding variables to the system. Most turbulence models consider that it can be represented by an increase in viscosity at each point of the domain. This depends on variables like turbulence kinetic energy and its dissipation rate. The turbulence model used in the simulations was K-Epsilon, which is more used in computational fluid mechanics problems. Due to mesh complexity and geometric size, it became inviable to use more refined turbulence models, like SST K-Omega or Reynolds Stress, which are more necessary in the prediction of flow with great rotational components and in boundary layer effects close to curved surfaces. Menter (1994) and Schaffarczyk (1999) carried out a detailed study of the use of these models.

2. MATERIALS AND METHODS

A horizontal wind tunnel was used to investigate the flow produced by an axial fan and a spraying system, determining its characteristic curves. The tests conducted in that installation provided quantitative results for pressure, flow and power. Assembly of the tunnel and measurement of the experimental results are according to the norm for laboratory assays (ANSI/AMCA, 1999). Figure 1 shows the schematic drawing for the test section with 622 mm in diameter.

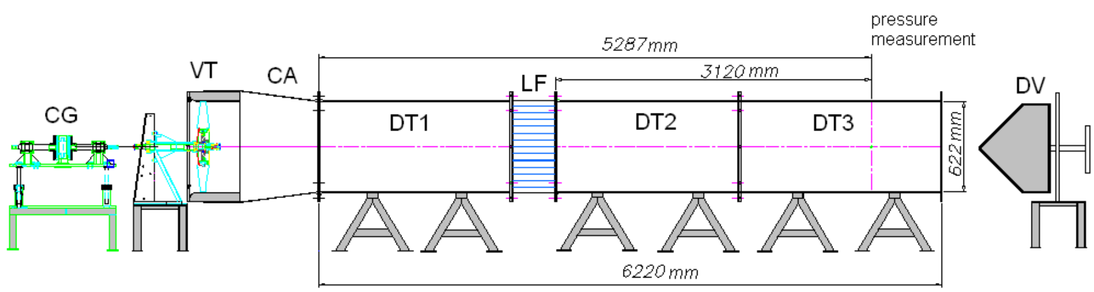


Figure 1. Schematic drawing of the test section

The test section is composed of the following components:
VT - Axial fan with 622 mm in diameter;
CA - Fan coupling box;
DT - Tubes with 622 mm in diameter;
LF - Flow laminator;
DV -Device of flow control;
CG - Load cell (torque meter).

2.1. Experimental procedure

The characteristic curves of the fan and the system were obtained with a 32 degree blade attack angle at rotations of 1500, 1750, 2600 and 3000 rpm.

2.1.1. Fan curves

Ten pressure measures were taken (using the Pitot tube) in the test section area of measurement for each rotation, adjusted with aid of a tachometer. Each measure was taken in a certain flow that can be adjusted with the device of flow control. This device was positioned at the exit of test section air and the first measure of pressure was taken. The device was then moved away 50, 100, 150, 200, 250, 300, 450 and 600 mm and for each position a new measure was taken. A measure was also taken with the test section open (without the device). A torque value was obtained for each flow in the axis of the fan that was monitored through the load cell.

With these measurement parameters it was possible to draw pressure curves, efficiency and power for the fan at four rotations.

2.1.2. System curves

The application system (Fig. 2 (b) and (c)) of the sprayer (Fig. 2 (a)) was coupled at the exit of the test section and then four pressure measures were taken in the test section measurement area, varying fan rotation. The characteristic curve for the spraying system was then plotted using the pressure values.

(a) Sprayer (b) Application system - lateral view (c) Application system - Front view

Figure 2. Sprayer and application system

3. MATHEMATICAL MODELLING

3.1. Numeric simulation

For real flow, a numeric treatment should be adopted, so the differential equations can be solved using a numeric method. This is because for Navier Stokes Equations, analytic solutions only exist for simple flows under ideal conditions (Bird, 1960). Numerical finite difference methods, and more recently, finite volumes, which are being used in this study, adapted easily to logical computational data treatment. The finite volume method is described thoroughly in literature in papers like Patankar (1980), Maliska (1995) and Versteeg & Malasekera (1995). Its application is discussed in recent such papers such as Ximenes (2004), Moreira (2002) and Fudihara (2000). The authors address topics such as numeric discretization and the consequent obtaining of linear equations and methods for solving resulting equations.

3.1.1. Transport equations

Approximate differential equations for an algebraic expression involve a discretization of the equations in the spatial domain inside a control volume. The governing equations are integrated in each control volume, so the important quantity (mass, momentum, energy, etc.) is conserved for each control volume.

In an isothermal and incompressible analysis (considered in this paper) the resolved equations for the discretized flow domain are the conservation of mass (Eq. (1)) and momentum (Eq. (2)) equations:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j\right)=0 \tag{1}$$

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

where ρ is the specific mass of the fluid, t is time, Ui and Uj are vector speeds, xi and xj are space components in Cartesian coordinates, P is pressure, μ_{ef} is the fluid's actual viscosity and fi is related to field forces.

3.2. Computational model

With 3D geometry, it was possible to discretize the structure in finite volumes using an unstructured tetrahedral mesh built with the ICEM CFD 5.0 mesh generator.

The model for analyzing the fan coupled in the tube was reduced to a 40 degree slice (blade angle) using a periodicity interface to represent the fan. This technique is very used to reduce the number of elements and processing time. This model is composed of three subdomains: a stationary subdomain before the fan (1), a rotational subdomain containing the fan (2) and a stationary subdomain after the fan (3) as presented in Fig. 3.

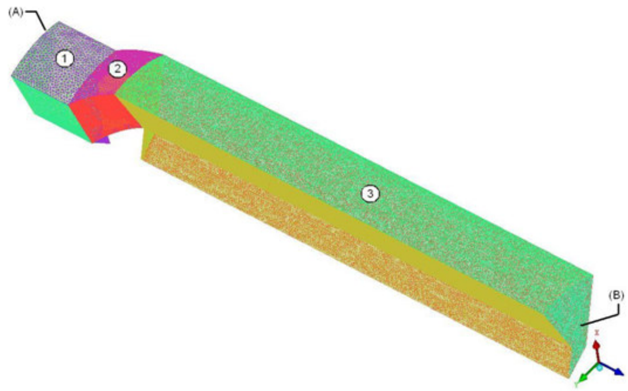


Figure 3. Details of the rotational domain

Volume for this model was considered to be around 1 million. Average residue among iterations under 10^{-4} was used as a convergence criterion. Specific air mass is considered constant in all simulations: (ρ = 1,0866 kg/m^3 and μ = 1,831.10^{-5} Pa.s), because the flow is considered incompressible and isothermal and gravity action is disregarded.

Two analysis types using CFX software were needed to determine the fan's characteristic curves: the first with the following contour conditions:

- Entrance of air with total relative pressure of zero (Inlet(A));
- Exit of air with total relative pressure of zero (opening (B));
- Speed of zero in stationary domain walls (condition of flat wall without slipping);
- Speed same as fan rotation for blade walls and cube in the rotational domain.

With these contour conditions, flow is produced by blade movement that is in the rotational subdomain in relation to the stationary domain and flow value at the exit is maximum.

In the second analysis the contour condition at the exit is altered for a flow value 10% inferior to the maximum value obtained in the first analysis. This procedure is repeated three more times, reducing the maximum flow value by 20, 30 and 40%. This provides five points for constructing the curves of the fan at each rotation.

The application system model is composed of three subdomains: a rotational subdomain containing the reduced fan at 40 degrees (1), a stationary subdomain with the fins at 45 degrees (angle for a fin (2)) and another stationary subdomain representing the exit of the application system (3) as presented in Fig. 4. Volume for this model was considered at around 3.5 million. For obtaining the system's curve several analyses were made varying fan rotation at 1500, 1750, 2600 and 3000 rpm with fixed contour conditions:

- Entrance of air with total relative pressure of zero (Inlet(A));
- Exit of air with total relative pressure of zero (opening (B));
- Speed of zero in stationary domain walls (condition of flat wall without slipping);
- Speed same as fan rotation for blade walls and cube in the rotational domain.

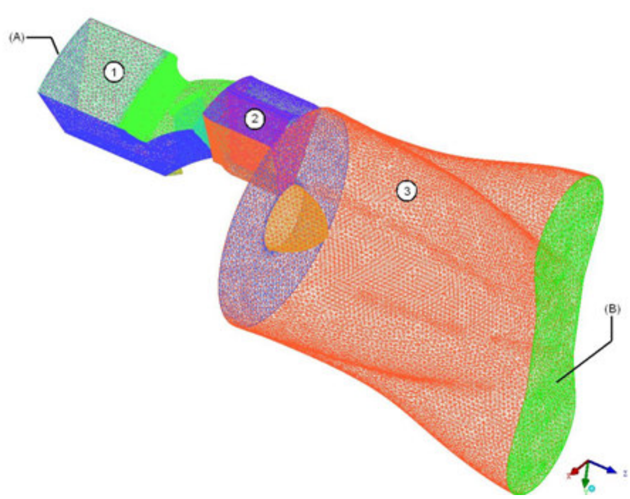


Figure 4. Application system calculation domain

4. RESULTS AND DISCUSSIONS

Experimentally obtained results were compared with those obtained numerically in form of characteristic curves for a 32 degree blade attack angle at 1500, 1750, 2600 and 3000 rpm. A numerical verification of the influence of blade attack angle variation was also done at 28, 32 and 36 degrees and of application system optimization, both at a rotation of 2600 rpm.

4.1. Experimental and numerical characteristic curves

A graph was generated for each rotation of the fan with the numerical and experimental results indicating the operation point in the application system curve.

Figure 5 shows the graph with efficiency, pressure and power curves for the fan at a rotation of 1500 rpm and the application system curves.

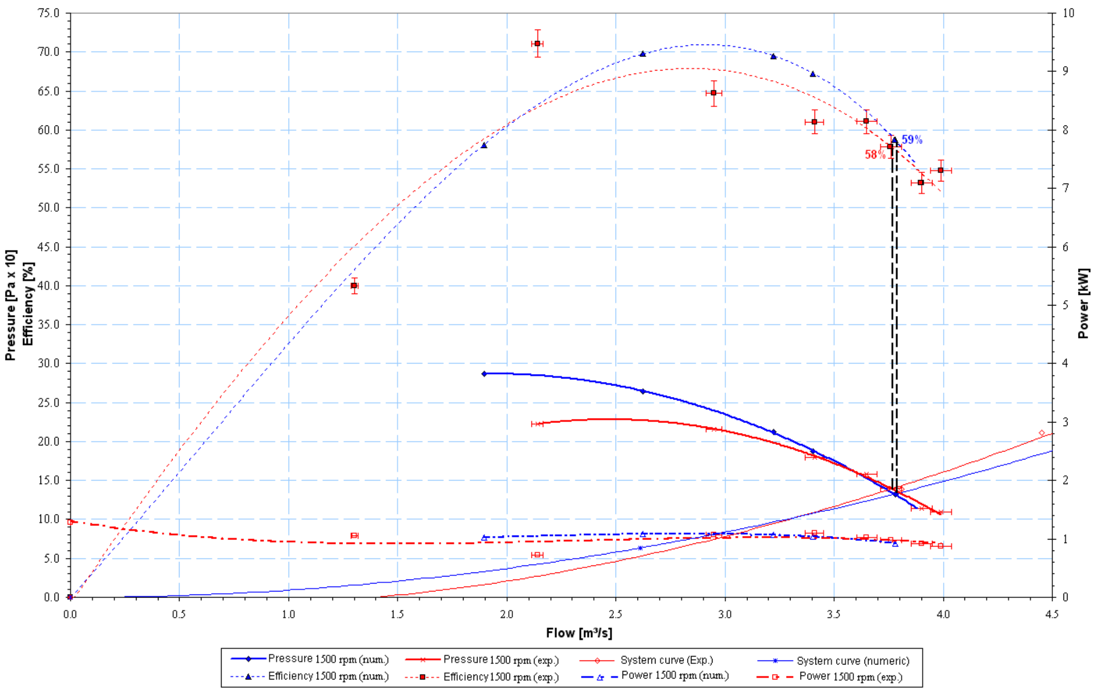


Figure 5. Numerical and experimental curves at a rotation of 1500 rpm

Experimental results show a fan efficiency in the application system of 58%. When comparing with numerical results, we see that all curves follow the same tendency, presenting system efficiency of 59%. The flow difference for the operation point in the two cases is very small, close to 0.5%.

Figure 6 presents the graph with the efficiency, pressure and power curves for the fan at a rotation of 1750 rpm and the curves for the application system.

System efficiency of 60.5% was obtained experimentally and numerical results show an efficiency of 59.5%. The operation point of the fan in the system in both cases coincides with a flow of approximately 4.4 m^3/s.

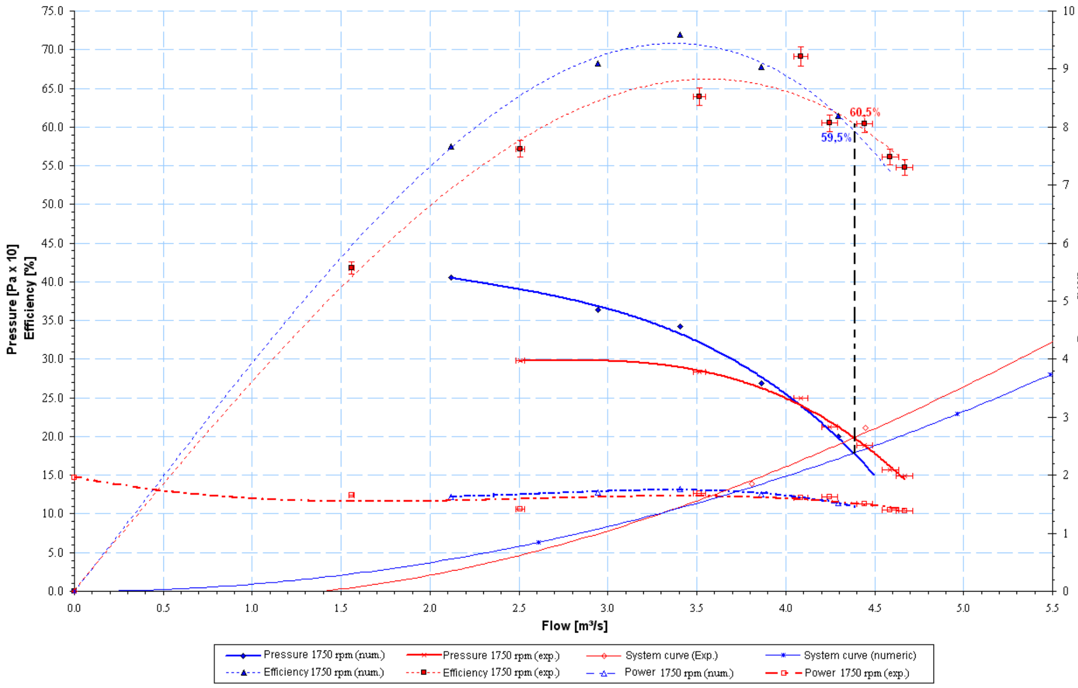


Figure 6. Numerical and experimental curves for a rotation of 1750 rpm

Figure 7 shows the graph with the efficiency, pressure and power curves of the fan at a rotation of 2600 rpm and the curves for the application system.

The fan operating at this rotation had the largest discrepancy among numerical-experimental results, a difference of 6.5% in efficiency, with 58.5% in numerical results and 65% in experimental ones. Flow difference at operation points is 3%.

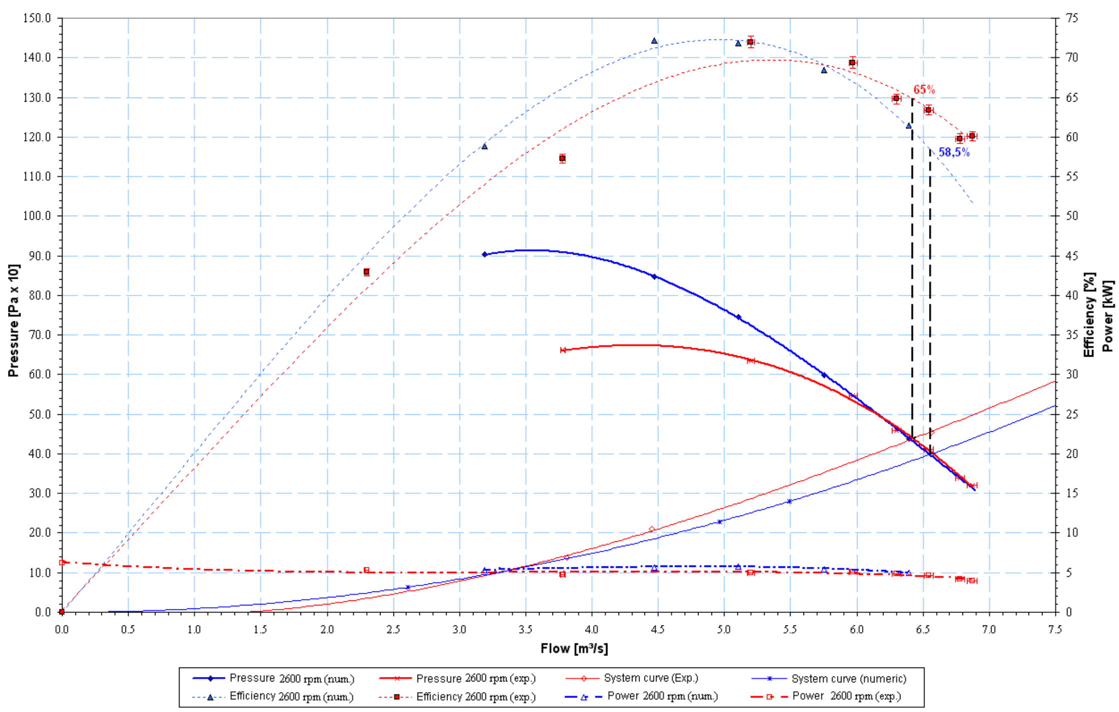


Figure 7. Numerical and experimental curves for a rotation of 2600 rpm

Figure 8 shows the graph with the efficiency, pressure and power curves of the fan at a rotation of 3000 rpm and the curves for the application system.

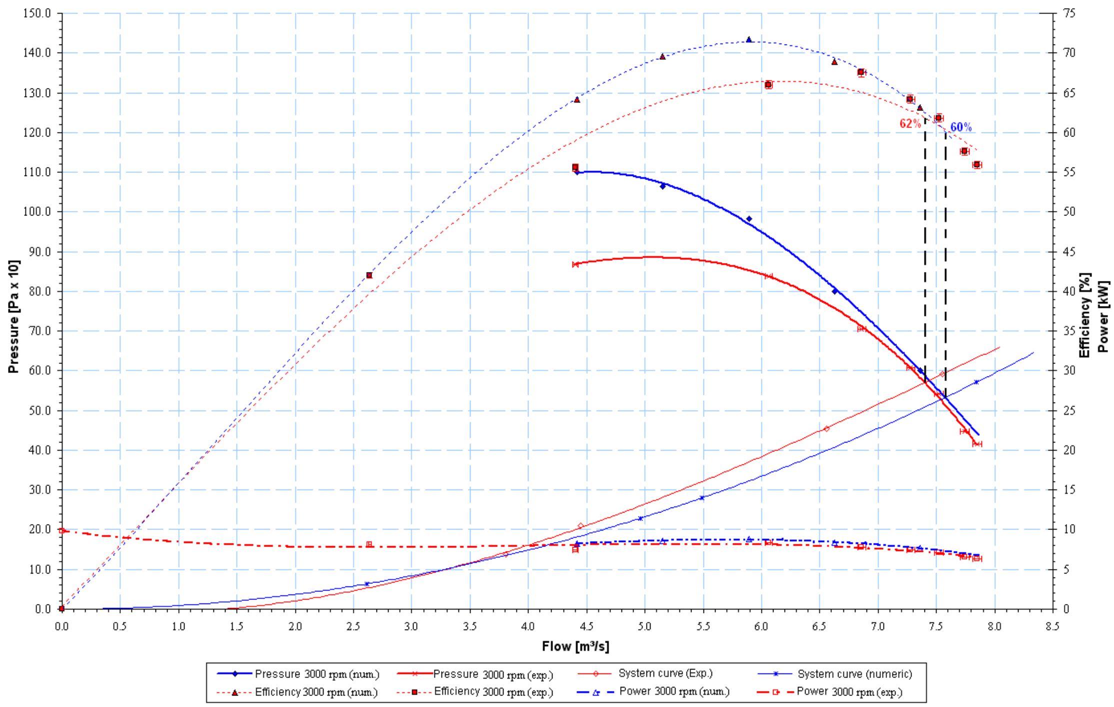


Figure 8. Numerical and experimental curves for a rotation of 3000 rpm

The difference in system efficiency between numerical and experimental results is 2%. At the operation point, the flow obtained numerically is close to 7.4 m^3/s and the experimental around 7.6 m^3/s, a difference of 2.7%.

Table 1 shows the difference between numerical and experimental efficiency at the operation point of the fan in the application system for each rotation.

Table 1. Numerical and experimental efficiency for the operation points

	Efficiency [%]			
	Rotation			
Results	*1500 rpm*	*1750 rpm*	*2600 rpm*	*3000 rpm*
Numerical	59.0	59.5	58.5	60.0
Experimental	58.0	60.5	65.0	62.0

Experimentally the rotation with the greatest efficiency is 2600 rpm (65%). The numerical results indicate a rotation of 3000 rpm (60%) as the most efficient.

4.2 Influence of the attack angle on the system

Verification of the blade attack angle consists of comparing the fan efficiency curves in the application system at a fixed rotation of 2600 rpm for angles at 28, 32 and 36 degrees.

Figure 9 presents the graph with the efficiency and pressure curves for the fan at a rotation of 2600 rpm and the curve for the application system.

Analyzing the points of operation for the application system, the blade with a 36-degree attack angle provides the greatest efficiency among verified angles, around 60%. When analyzing efficiency curves, we see that when the attack angle is increased from 32 degrees to 36 degrees, the point of maximum efficiency falls from 72 to 68%. Thus, depending on the system in operation, it is not appropriate to increase the attack angle.

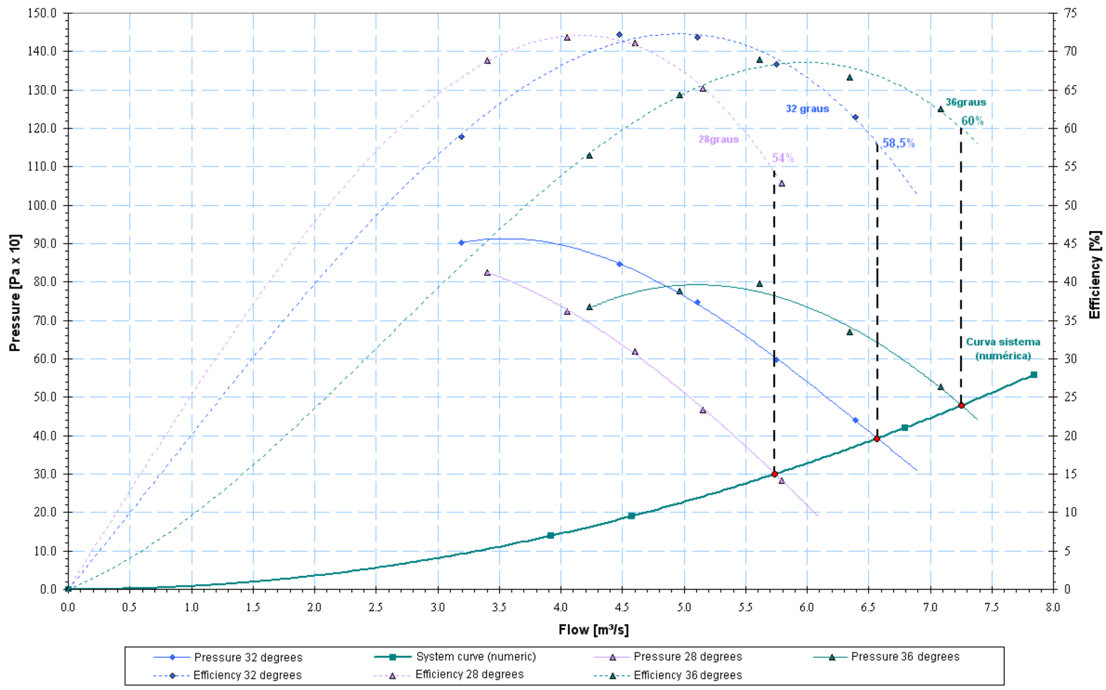
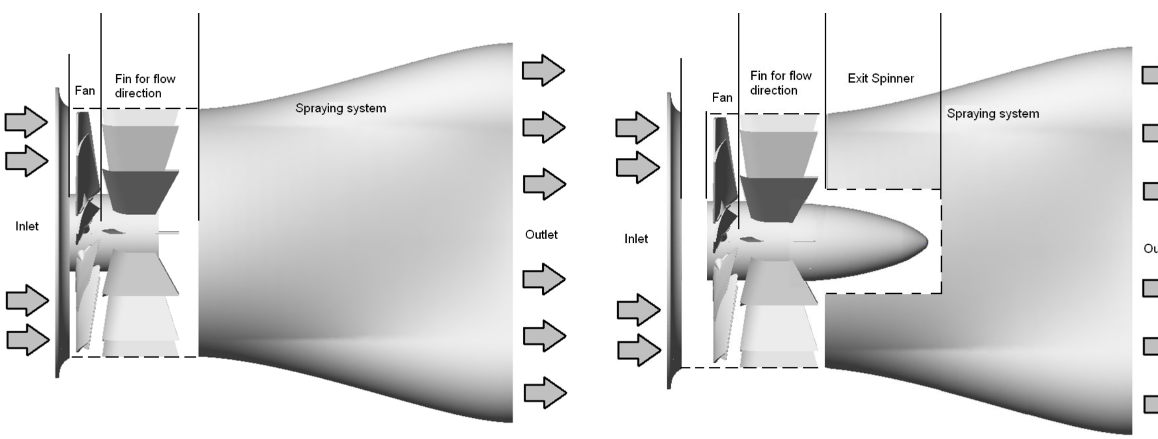


Figure. 9. Numerical curves for a rotation of 2600 rpm

4.3. Optimization of the application system

Two analyses of the application system were made: the first in the initial configuration according to Fig. 10 (a) and second with the optimized model moving the fan and fins 80 mm into the application system and inserting a spinner in front of the fins as shown in Fig. 10 (b).



(a) Initial configuration (b) Optimized model

Figure 10. Models of the application system

Table 2 shows the pressure, efficiency, flow and power results of the fan entrance for the two configurations.

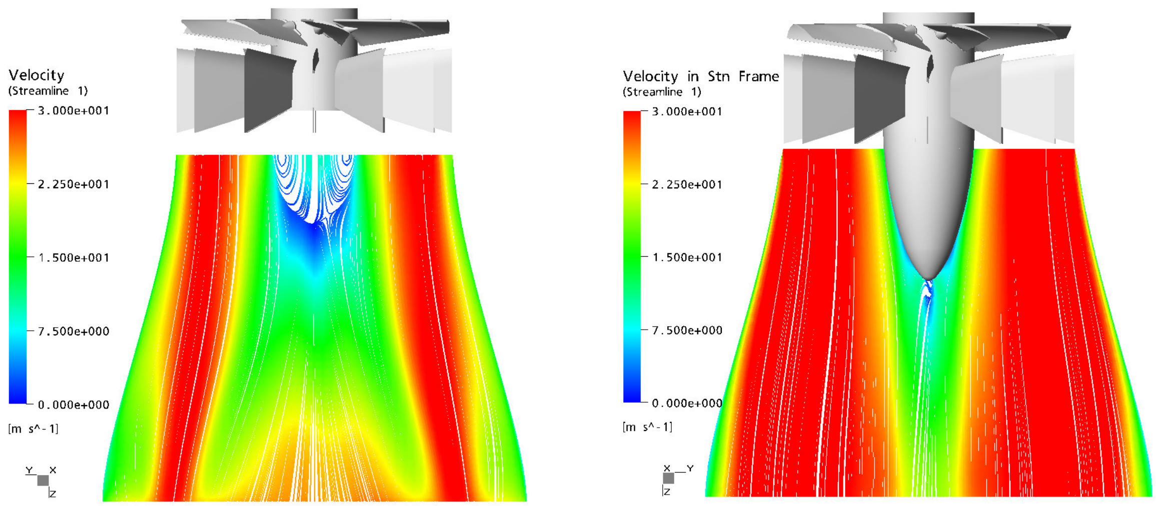
Table 2. Comparative numerical results

	Calculated parameters			
Models	Power inlet [W]	Flow [m^3/s]	Pressure [Pa]	Efficiency [%]
Initial configuration	5085.8	6.67	400.59	57.18
Optimized model	4342.4	6.21	397.16	61.96

We see that the efficiency calculated for the initial model is 57.18% and the value obtained by the graph of the characteristic curves for this same rotation of 2600 rpm is around 58.50% (Fig. 7). This difference is related to the approach of the curves, since they were generated starting from points and approximated for a third order polynomial trend line.

Comparing the optimized with the initial model we see there was an increase of around 5% in efficiency. This happens due to the decrease in recirculation after the fins providing a decrease in fan torque.

Figure 11 presents the velocity profile in the central plane of the application system in streamlines.

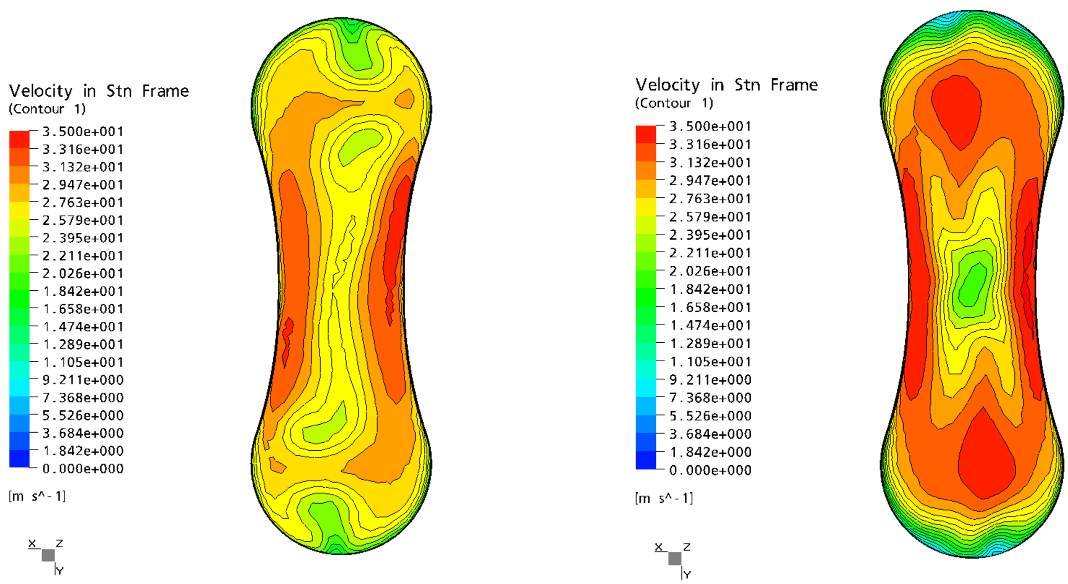


(a) Initial configuration

(b) Optimized model

Figure 11. Streamlines in application system

Analyzing Fig. 12, we see that the model with the spinner also has improved velocity distribution at the exit of the application system with smaller areas of low velocity at the extremities.



(a) Initial configuration

(b) Optimized model

Figure 12. Velocity gradient in the application system

It was verified that the presence of the spinner and the displacement of the rotor into the application system increase efficiency reducing recirculation effects at the extremity of the blade and after the fins.

5. CONCLUSIONS

CFX software presented reliable results, close to those measured experimentally. Through mesh choice parameters, turbulence models and appropriate advection schemes we can arrive at models capable of numerically projecting the behavior of a flow machine, becoming a tool of great importance in your project. Use of this simulation in a company can reduce test costs and rework, first making simulations with several models and later just building the most appropriate for experimental tests. Besides, visualization of the flow is an important point for analysis by the company's technical team. It was also possible to demonstrate the usefulness of CFD methods for the company's advanced engineering project and for defining an appropriate methodology for mesh generation and appropriate turbulence models. For this study, the K-Epsilon model was sufficiently satisfactory for the general profile of the flow.

6. REFERENCES

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

Bird, R. B., Stewart, W. E. & Lightfoot, E. N., 1960, "Transport Phenomena" John Wiley & Sons, pp. 780.

Fudihara, T. J., 2000, "Método dos Volumes Finitos aplicado à modelagem matemática e simulação computacional de um forno aquecido por um jato de chama com escoamento em vórtice", Tese de doutorado, FEQ/UNICAMP, Campinas, 165p.

Kelecy, F., 2000, "Study Demonstrates that Simulation Can Accurately Predict Fan Performance", Journal Articles by Fluent Software Users, New Hampshire- USA.

Maliska, C. R., 1995, "Transferência de Calor e Mecânica dos Fluidos Computacional: Fundamentos e Coordenadas Generalizadas", Rio de Janeiro: LTC.

Menter, F.R., 1994, "Two equation Eddy Viscosity Turbulence Models for Engineering Applications", American Institute of Aeronautics and Astronautics Journal, 32(8).

Moreira, D. R. R., 2002, "Simulação não isotérmica de um regenerador usando a fluido dinâmica computacional", Dissertação de mestrado, FEQ/UNICAMP, Campinas, 92p.

Patankar, S. V., 1980, "Numerical Heat Transfer and Fluid Flow", Ed. MacGraw-Hill, New York, 193 p.

Schaffarczyk, A. P., 1999, "Prediction of Airfoil Characteristics for Wind Turbine Blades with CFX". Laboratory of Computational Mechanics, University of Applied Sciences, Kiel – Germany.

Versteeg, H. K. & Malasekera, W., 1995, "An Introduction to Computational Fluid Dynamics - The Finite Volume Method", England: Longmsn Group Ltda.

Ximenes, C. S., 2004, "Aplicação de Técnicas de Fluidodinâmica Computacional (CFD) em Fornos para Produção de Cimento", Dissertação de mestrado, FEQ/UNICAMP, Campinas, 146p.

7. RESPONSIBILITY NOTICE

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