Developing an industrial centrifugal fan as prototype using an experiment series and finite volume method

K. Turgut GÜRSEL, Onur GÜRSEL, Mehmet ERKEK
Mechanical Engineering Department, Faculty of Engineering, Ege University, 35100 Bornova – Izmir / Turkey
Phone and fax: +90 232 388 8562; turgut.gursel@ege.edu.tr; mehmet.erkek@ege.edu.tr

Abstract— Centrifugal fans are used in most of the manufacturing processes for ventilating and/or air conditioning the manufacturing areas. Because of relatively limited documentation on design of these fans, new designs are developed by experimental method. This method does not only take a lot of time but also increases the costs considerably. Nowadays, most of the companies create 3D models and then conduct analyses by the CFD (computational fluid dynamics) programs and perform some optimizations before manufacturing their new designs.

In this study, the main principles of centrifugal fans and their characteristics were treated. Further, a fan was developed and a prototype of it was manufactured. Then the steps of its numerical and experimental performance tests and the comparison of these results were accomplished.

***Index Terms*— Industrial centrifugal fans, performance test, finite volume method (FVM), computational fluid dynamics (CFD).**

I. INTRODUCTION

Various types of fans are used in different branches of industry, houses and buildings for purposes such as air conditioning, cooling, heat recovery and air supply for industrial processes. Having so many different applications, each fan type needs detailed study. In this study, industrial centrifugal fans were treated, and one was developed by means of different considerations and analyses.

In all industrial manufacturing facilities, at least one fan is used either for the supply of process air or ventilating. The working principles of fans are based on the fundamentals of fluid mechanics, and there is a range of CFD (computational fluid dynamics) software on this subject. In the calculation and selection methods as well as design of these fans, the number of documents and information on these subjects used by engineers or other technical personnel can be limited under certain conditions. For this reason big mistakes leading to loss of energy can be made in the selection of fans. The correct selection and operation of fans that are so widely used, give the enterprise a chance to use these fans continuously and economically.

As a result of this, most of the fan designs are made based on experience and these are tested afterwards. If these tests do not produce reasonable results, the design must be reconsidered. When the manufacturing and testing of the prototypes are considered, this process is quite an expensive and time consuming method. Nowadays the electricity consumption of fans is still an important parameter. Therefore the necessity of better designs is increasing. Nowadays many types of fans exist and have very wide use. But when it comes to industrial centrifugal fans there have been certain studies.

Jeon and Lee [1] conducted studies on the effects of fan design criteria on performance and noise. There are two sources of noise in fans which are shaft/rotor vibration and air flow. In this study the effects of design parameters on noise occurrence due to air flow were investigated. This noise is caused by aerodynamic forces and motions during air flow and is difficult to predict due to the irregular flow patterns in fans. In their study John and Lee developed a numerical method based on some assumptions. They analyzed and determined the noise that can be expected at various values of the design parameters.

Lin and Huang [2] performed numerical and experimental studies on inclined centrifugal fans used in notebooks for cooling. In this study results obtained by CFD analysis and experimentation were compared. Finally they determined that the fan proved sufficient performance by optimizations carried out.

Eck [3] mentions about fan types, their areas of use, fan tests and characteristic curves in his book called “Fan Handbook”.

Arun and Akkoc [4] deal with the design of lines used to transfer particles in the form of powder in the book called “Fundamentals of Pneumatic Transfer”. Here they give information about the equipments including industrial centrifugal fans used in these lines.

Bohl [5] gives detailed information on fans in his book “Ventilator” in which fan characteristics and their changes are treated.

Ugural and Parmaksızoglu [6] give information about calculations of ventilation ducts, duct losses, fan characteristic curves, basic laws of fans and electrical motors.

Information on the necessary equipments for fan tests, the test stand and calculation methods for these tests are given in AMCA 210-99 (1999) [7].

Fujii and Tamura [8] conducted a study about the capacity of computers in the solution of the equations resulting from the application of basic fluid mechanics principals in computational fluid dynamics.

Engin [9] carried out a three-dimensional CFD simulation of the flow field in three different unshrouded centrifugal fan impellers with varying tip clearances and used therefore a commercial CFD code with a k-ε two-equation turbulence model in order to study the effects of tip clearance on the overall performance of each fan. The numerical results were compared with the experimental data reported previously in the literature.

Zhou et al. [10] executed a three-dimensional simulation of internal flow in three different centrifugal pumps using the CFD, and Asuaje et al. [11],[12] developed inverse design method for centrifugal impellers and accomplished two codes that could be used for the design and performance analysis of both centrifugal and mixed flow pumps.

Yu et al. [13] studied the effects of impeller geometry on the performance of a centrifugal blood pump model both experimentally and computationally, using the CFD and Yu et al. [14] investigated numerically the flow field by considering blade inlet angle and tip clearance as the key parameters.

II. AIM OF THE STUDY AND METHOD

In many stages of production processes, high flow rate air necessary either as an input for the processes or for ventilating purposes is supplied by industrial centrifugal fans.

Fan performance that is known as fan characteristic curves is of major importance during the constant speed operation of industrial centrifugal fans. These curves give information about the relation between the air pressure and the flow rate and show the effects of the system characteristics on the performance of the fan.

The characteristic curves are obtained by the data from experiments conducted after the production of the fan and cannot be expressed by empirical formulas. Therefore for each new design a prototype must be produced and experiments must be performed on this prototype for determining the performance of the new design. This is a time consuming and expensive procedure. Additionally it is possible that the design obtained may not be optimum.

In this study a computer model for an industrial centrifugal fan was developed and its performance was analyzed using FLUENT software. The prototype of the developed model was manufactured and tested on a test stand complying with the standards and the results of both experiment and numeric analysis were compared. The aim was to come as close to the experimental results by the analyses so that computer software can be better utilized as a tool in design. By this way developing industrial centrifugal fans and/or redesigns can be first tested by using this software before producing the prototype which can prove to be useful in reducing the development/design costs.

III. INDUSTRIAL CENTRIFUGAL FANS AND THEIR CHARACTERISTICS

According to Euler's fundamental equation concerning fluid machinery, the pressure difference that a centrifugal fan delivers is given by equation 1.

$$\Delta p = \eta_h \rho_H \left(w_2 C_{u2} - w_1 C_{u1} \right) \qquad (1)$$

In Eq. (1) the subscripts (1) and (2) denote the inlet and outlet conditions, respectively. η_h is an efficiency coefficient, and w_1, w_2 and C_{u1}, C_{u2} are peripheral speeds of fan rotor and fluid respectively. If the inlet velocities are neglected and if we use the simplifications of $w_2 \approx C_{u2}$, equation (1) can be rearranged in the form of equation (2). The efficiency coefficient η_h can be taken between 0.6-0.8.

$$\Delta p = \eta_h \rho_H w_2^2 \quad (\mathrm{N/m^2}) \qquad (2)$$

According to EN ISO 5801:2008 [15] standards the following distinctions are made between fans and compressors (EN ISO 5801:2008 Part 1: Ventilatoren-Leistungsmessungen, Normkennlinien) [5], [15], [16]:

For fans: $p_1/p_2 < 1.3$

For compressors: $p_1/p_2 > 1.3$

The pressure produced by fans decreases proportional to the density of the air ρ_H at the suction line. High flow rates of the industrial fans bring about high noise levels. In order to prevent this, acoustic insulation and/or use of silencer must be employed.

A. Characteristics of fans

A selected fan must be able to deliver air at the necessary flow rate while overcoming the pressure resistance of the equipments of the system in which this fan was installed. If the fan is designed so as to work at optimum regime in its design conditions, it can work at reasonably high efficiencies for a large pressure-flow rate range. Fans' performances are determined by a diagram which shows the relation between the flow rate and pressure delivered by the fan and this diagram is called the "fan characteristic curve". The point where the fan's characteristic curve and the system's characteristic one intersects is the operating point of that fan (Figure 1). System characteristic curve can be defined by equation (3) in which the constant K denotes the pressure losses in the system.

$$\Delta p = KQ^2 \quad (3)$$

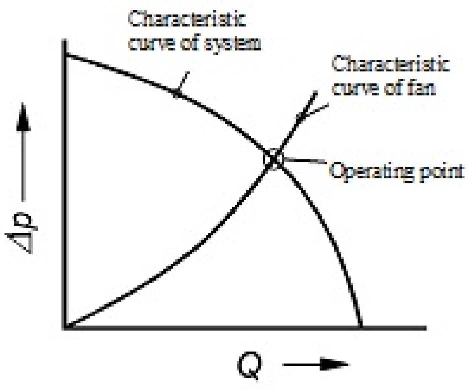


Fig 1. Characteristic Curve of a Fan as well as operating point

There is a certain range where each fan type can be used due to its properties related to the construction and capacity of fans. Within this working range, some alterations can be made at the characteristic curve and operating point of fans by various methods.

The first method to be applied is to use a variable speed frequency converter to shift the fan's characteristic curve up and down within the working region. By this way the operating point can be shifted to points where the fan possesses optimum efficiency when system rotational speed needs to be changed for various reasons (Figure 2). Although this method has a high initial cost it provides energy savings where system resistance shows variations.

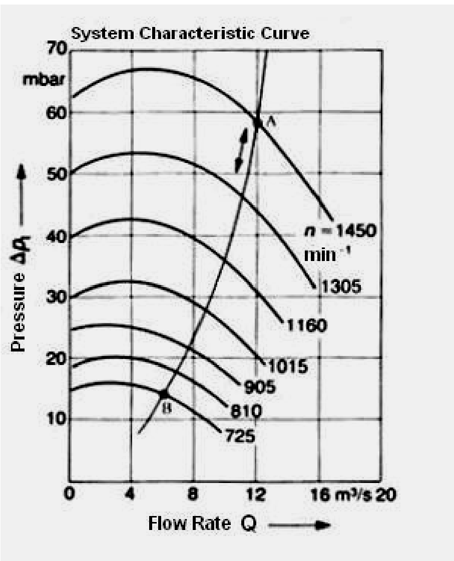


Fig 2. Characteristic Operating Range of a Centrifugal Fan with speed control.

The second method is to use a valve at the fan's suction line. This increases the system resistance without changing the fan characteristics and changes the operating point. By closing the valve, an additional resistance is created in the system which increases the system characteristic curve's gradient.

The final method used to change fan characteristics is to place vane-valve at the suction side of fans (Figure 3). At different positions of this valve different fan characteristics can be obtained. Closing the valve causes rise of eddies in the flow coming towards the radial fan wings which makes the characteristic curve steeper. The operating point also changes due to the change in the characteristic curve. The change in the characteristic curve of a fan at various positions of the valve is given in Figure 4. The power needed for a fan using this method will be more when compared with that of a fan using variable speed method.

Fig 3 Vane Valve

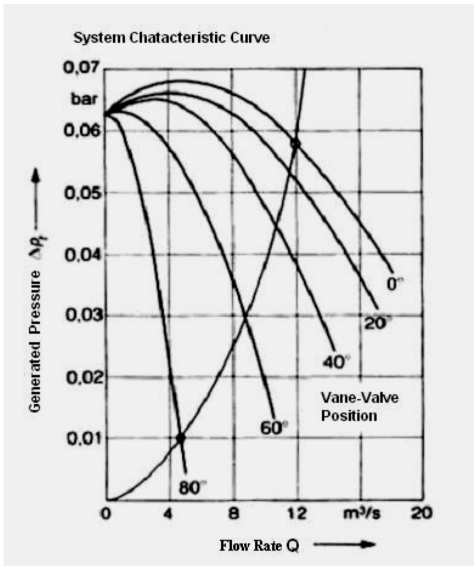


Fig 4. Characteristic Curve of a Centrifugal Fan at constant speed adjusted by a vane-valve.

B. Selection of the Fan and Principal Calculations

The first step in the selection of an industrial fan is to determine properties of the system that is going to use the fan and requirements of the fan correctly. Fan selection can be analyzed in two phases namely determination of fan performance and system conditions.

I. The first step in selecting a fan is to determine the necessary air flow rate and pressure for the system. Flow rate is the necessary volume of air in unit time for the system and can be found by multiplying the air velocity in the cross section of the system with the area of that cross section. The necessary pressure for the system is the sum of the pressure losses in the ducts of the system through which the fan will suck or push the air and the pressure losses occurring in equipment along the ducts such as filters, cyclones, furnaces, etc.

II. In addition, the temperature of the air that is going to flow through the fan and the altitude of the place where the fan will work must be taken into account. The altitude and temperature will change the density of the air. Therefore if we use the fan diagrams prepared for air under normal conditions (at 20º C at sea level) this will cause a significant error. Table of air density ratio gives the effect of air density on fan performance and correction coefficients [15].

In selection of industrial fans, it is very important to correctly determine the flow rate and pressure reduced to normal conditions in order to obtain correct characteristics for the fan. The next step is to correctly determine the conditions of the system and environment in which the selected fan will be used. This directly affects the fan construction. Therefore the following points must be considered here:

- Properties of the air - passing through the fan - such as its humidity, temperature, composition, and whether it is corrosive, contains flammable gases, all effect the material choice for the fan. It may also be necessary to take special precautions against explosions.
- The allowable noise level where the fan will be used must be determined. After this it might be necessary to use noise insulation.
- It is necessary to determine the power needed to operate the fan. Generally the necessary power is supplied by an electric motor and the following points must be clarified when using an electric motor:
  - Whether the electricity source is AC or DC,
  - If the electricity source is AC, then whether it is single or three phased,
  - Finally the voltage must be determined.
- Finally the direction of the air ventilation and the position of the fan in the assembly area have to be determined.

After the necessary surveys for selecting the fan are conducted, the necessary equations (4-7) related to the calculation of flow rate, pressure and power of the fan are given as follows:

$$\sum p_T = p_{total,out} - p_{total,in} \quad (4)$$

$$\sum p_T = p_{st_{out}} + p_{dyn_{out}} - \left[p_{st_{in}} + p_{dyn_{in}}\right] \quad (5)$$

$$Q(m^3/h) = A(m^2)\cdot u(m/s)\cdot 3600\,(\text{s/h}) \quad (6)$$

$$N_{shaft}(kW) = \frac{P_T \cdot Q}{3600\cdot 102\cdot \eta_f}. \quad (7)$$

Q = Air flow rate (m^3/h)
p_{st} = Static pressure (mm H_2O)
p_{dyn} = Dynamic pressure (mm H_2O)
p_T = Total pressure (mm H_2O)
N_{shaft} = Power used by the fan (kW)
η_f = Fan efficiency

C. *Effect of Air Density on Performance of the Fan*

The diagrams and tables of fans are prepared for air under normal conditions in which it possesses 1.2 kg/m^3 density at 20ºC and 760 mmHg. If the fan is working at an altitude of h (m) at T (ºC), the air density will change and attain a value of ρ_1. Under these conditions the following will be valid according to the laws of similarity for fans.

- The flow rate remains the same when the air density changes.
- The pressure delivered by the fan changes proportionally with the change of density.
- The power used by the fan changes proportionally with the change of density.

Table of air density ratio [15] displays “K” correction factor which is the ratio of the density of the air under normal conditions to the density of the air at an elevation of “h” and at a temperature of “T”. The fan selection criteria must appropriately be changed by taking into account the change of the air density in order to be able to use the fan selection diagrams prepared for normal conditions.

IV. DETERMINATION OF CHARACTERISTIC VALUES OF THE CENTRIFUGAL FAN USING FVM

A. *Use of Numerical Methods in Fluid Dynamics*

While fluid systems can be expressed by complicated differential equations, it becomes very difficult to solve these equations when especially 3D analysis is needed. For this reason many software using finite volume method to solve these complex or insolvable problems, at the necessary points within proper boundary conditions with a reasonable error are developed. So, computational fluid dynamics (CFD) has been formed by combining analytical fluid dynamics (AFD) and experimental fluid dynamics (EFD). The advantages of CFD are as follows:

- “Simulation and design” concept consisting of the phases of design, analysis and redesign is very useful, and it has eliminated the “*manufacture and test*” procedure.
- Complicated flow geometries are difficult systems for testing, measuring and controlling. It has simplified the simulation of very complex flow phenomena considerably.
- Natural parameters such as wind, sun etc. that can not be controlled during an experiment can be modified during simulation.
- It gives us the opportunity to simulate an experiment exactly by inputting experiment parameters into software.

- It gives the chance to get results by using simulation methods without being exposed to non-desired effects such as pollution, radiation, noise, etc. But still the results must be controlled via experimentation.

FLUENT is general purpose CFD software that is used to solve fluid mechanics and heat transfer problems occurring in very different industries like automotive, aeronautical, home appliances, turbo machinery (fans, compressors, pumps, turbines, etc.) and chemical and food industries.

B. Solution Procedure of FLUENT

FLUENT is a computational fluid dynamics software suitable for incompressible (low subsonic), medium compressible (transonic) and highly compressible (supersonic and hypersonic) flows [17]. With its numerical grid combinations that speed up convergence to the solution and through its solver options, FLUENT has an optimum solving efficiency and accuracy in a wide velocity range.

The "virtual geometry" concept available within GAMBIT simplifies the process of geometry cleaning which is the biggest problem encountered in imported models, and allows obtaining models more suitable for meshing.

FLUENT reads the "mesh" files transferred from GAMBIT or other preprocessors and solves the system according to the applied boundary conditions. FLUENT uses the equations (8-9) which are the fundamental equations of fluid mechanics, to solve the problem [8].

- Navier-Stokes Equation:

$$\underbrace{\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}}_{Convection\_terms}=\underbrace{-\frac{\partial p}{\partial x}+\mu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]+\rho g_x}_{Pressure\_gradient\_and\_viscosity\_terms} \quad (8a)$$

$$\underbrace{\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}}_{Convection\_terms}=\underbrace{-\frac{\partial p}{\partial y}+\mu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right]+\rho g_y}_{Pressure\_gradient\_and\_viscosity\_terms} \quad (8b)$$

- Continuity Equation:

$$\frac{\partial\rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \quad (9)$$

FLUENT uses finite volume method in order to solve these three fundamental equations at each volume element of the geometry in question. The main reason for using finite volume method in computational fluid dynamics is that the method enables flexibility in both geometry and physical phenomena parallel to the physical quantities. However the finite element method that is more widely used in structural analysis gives flexibility in the geometry and can be used with its general purpose codes for solution of different problems.

C. Modeling of Industrial Centrifugal Fan and Creating of its Mesh

The centrifugal fan to be analyzed was modeled by using the software "Unigraphics" and was transformed into a parasolid file. Then it was exported into GAMBIT in order to form the air volume in the fan and to mesh the whole geometry as seen in Figure 5-6. After this the next step is to choose the solver.

.

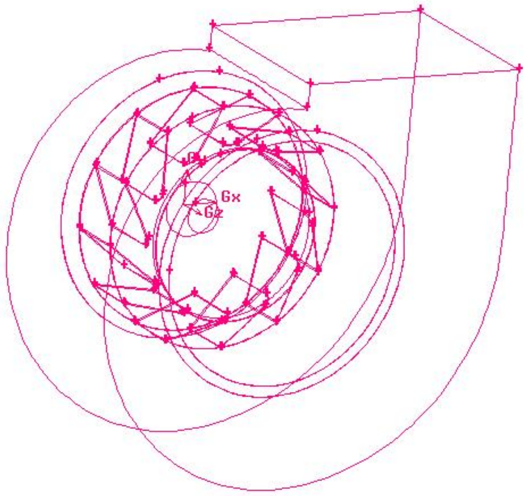


Figure 5. Model of the industrial centrifugal fan

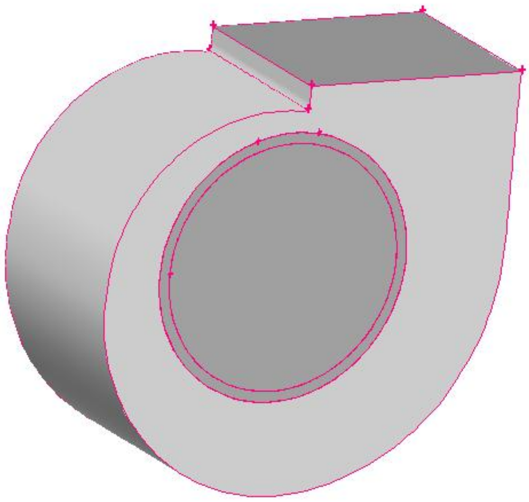


Figure 6. Model of the centrifugal fan whose air volume was created

Each solver has different properties and characteristics. GAMBIT determines the mesh type and boundary conditions to be defined according to the chosen solver. Because the types of mesh and boundary conditions are different for each solver, the most suitable solver for the investigated system must be chosen. In this study the FLUENT/UNS solver and finite volume method were used. The next step after the selection of the solver is to check the model geometry. This gives the possibility to spot any discontinuities at the edges and corners of the geometries, which may have resulted during transfer of models created using different software. This process is important because it prevents these discontinuities to cause larger problems during further steps of the analysis.

After the geometry checking and cleaning processes have been performed on the model, meshing that is the discretization of the geometry into a numerical grid is carried out. In the procedure followed during this study more elements were used near the volume swept by the rotor, and the element size increases away from this volume. Therefore, both the solution accuracy and time as well as processor capacity were taken into consideration. Dimension functions and "cooper mesh" were used as the numerical grid type (Fig. 7-8).

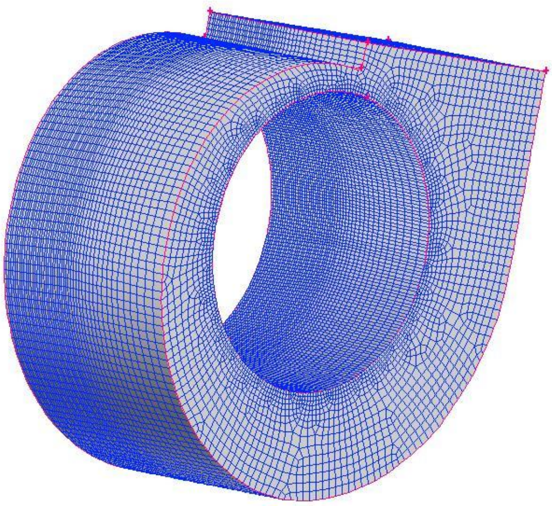


Figure 7. Housing of the meshed centrifugal fan

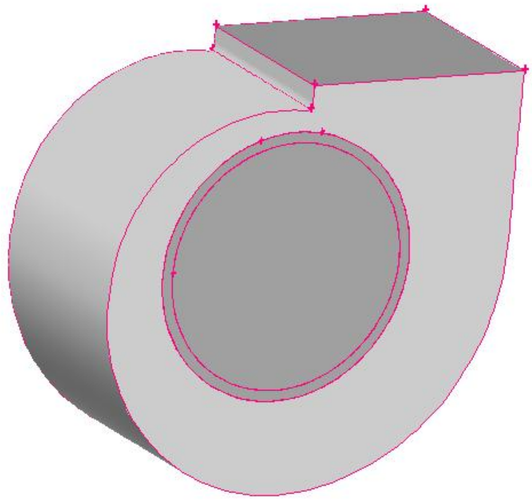


Figure 8. Meshed fan rotor and air volume

Because the analysis performed on the formed numerical mesh geometry directly affects the results, the number and the quality of the elements should be checked. In order to achieve grid independence, a total number of 1,245,059 elements was used in the mesh having consisted of 1,025,099 quad, 104,040 wedge and 115,920 hexagonal type elements. In GAMBIT the quality of these elements is characterized by a factor called "skewness factor" that has values between 0 and 1. It is best to have this factor as close as possible to zero. The percentage of the mesh elements having a "skewness factor" greater than 0.75 is given as 0.04 %, and Figure 9 exhibits their distribution. In Figures 10 and 11 the elements having a factor in the ranges of 0.6-0.75 and 0.3-0.6, respectively, and their distributions are analyzed. The number, skewness factors and distributions of the elements prove that the applied mesh quality is suitable for analyses and enables grid independence.

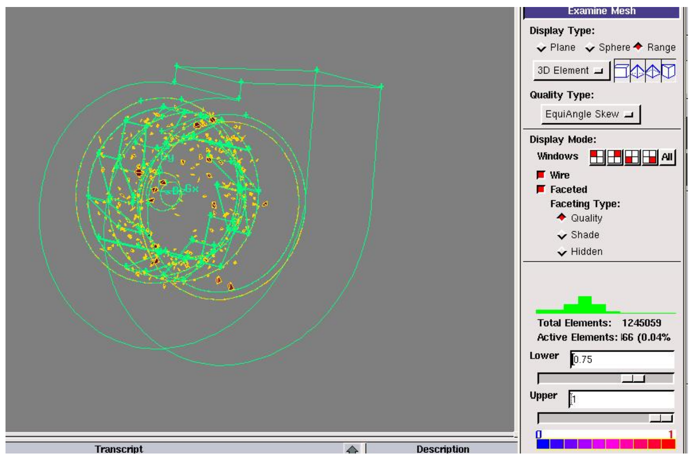


Figure 9 Mesh elements having skewness factor between 0.75-1

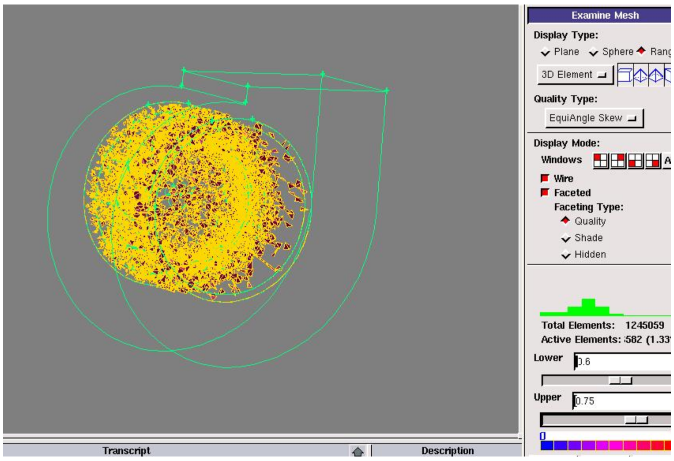


Figure 10 Mesh elements having skewness factor between 0.6-0.75

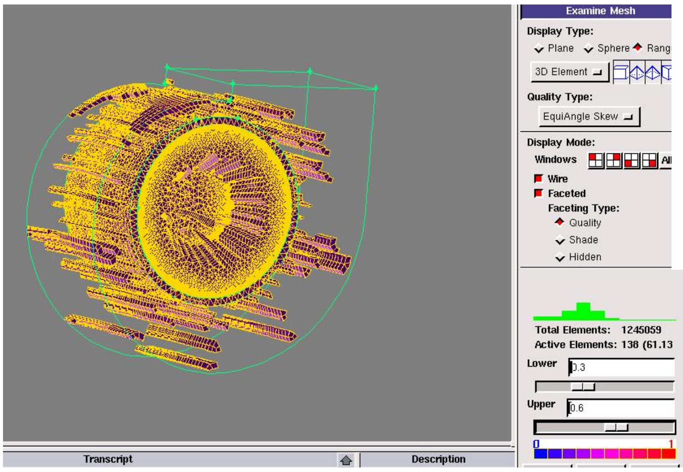


Figure 11 Mesh elements having skewness factor between 0.3-0.6

After checking the suitability of the mesh formed using GAMBIT, the boundary types of the modeled system were defined as wall for the solid volumes and fluid for the air volume and the moving air volume around the rotor, before transferring the model into FLUENT. After this the meshfile with "msh" extension was saved in order to ex port into FLUENT

D. *Solution of the Meshed Model in Fluent*

The mesh file with "msh" extension created on GAMBIT was transferred into and read by FLUENT. After this the following processes are performed:

- The first process to be performed on FLUENT was the "grid-check" process by which the mesh geometry read was checked. "Segregated solver" as solver type of the problem was chosen in this study. This solves each equation separately. Figure 12 exhibits the flow chart of this solver.

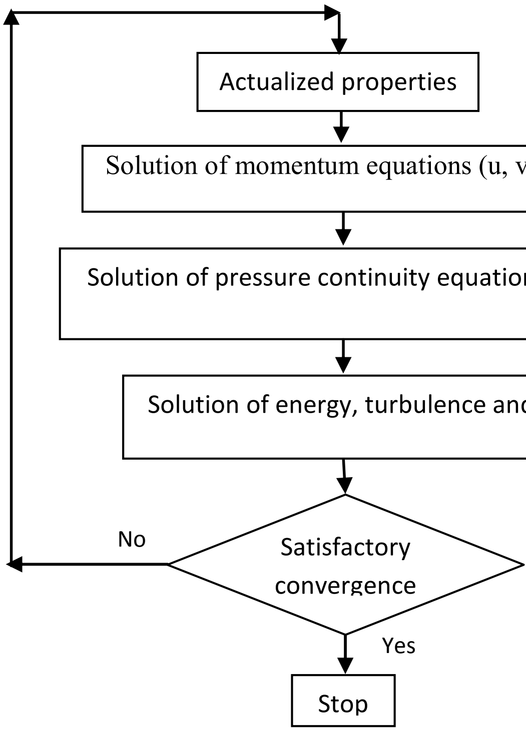


Figure 12. Flowchart of "segregated solver"

- In this study "standard k- ε model" was defined as the ıid model. In this model there are two equations. "k" is the uation obtained by subtracting the actual mechanical ergy equations from their average values in time. "ε" is : equation that involves the physical properties. This lution is suitable for all turbulent flows and delivers ısonably good results in industrial process flows and heat ınsfer problems.

Kinetic energy equation of turbulence (k):

$$\rho U_i \frac{\delta k}{\delta x_i} = \mu_i\left(\frac{\delta U_j}{\delta x_i}+\frac{\delta U_i}{\delta x_j}\right)\frac{\delta U_j}{\delta x_i} + \frac{\delta}{\delta x_i}\left\{\left(\mu_i/\sigma_k\right)\frac{\delta_k}{\delta x_i}\right\} - \rho\varepsilon \quad (10)$$

Equation of distribution ratio (ε):

$$\rho U_j \frac{\delta \varepsilon}{\delta x_i} = c_{1\varepsilon}\left(\frac{\varepsilon}{k}\right)\mu_i\left(\frac{\delta U_j}{\delta x_i}+\frac{\delta U_i}{\delta x_j}\right)\frac{\delta U_j}{\delta x_i} + \frac{\delta}{\delta x_i}\left\{\left(\mu_i/\sigma_\varepsilon\right)\frac{\delta_\varepsilon}{\delta x_i}\right\} - c_{2\varepsilon}\rho\left(\frac{\varepsilon^2}{k}\right) \quad (11)$$

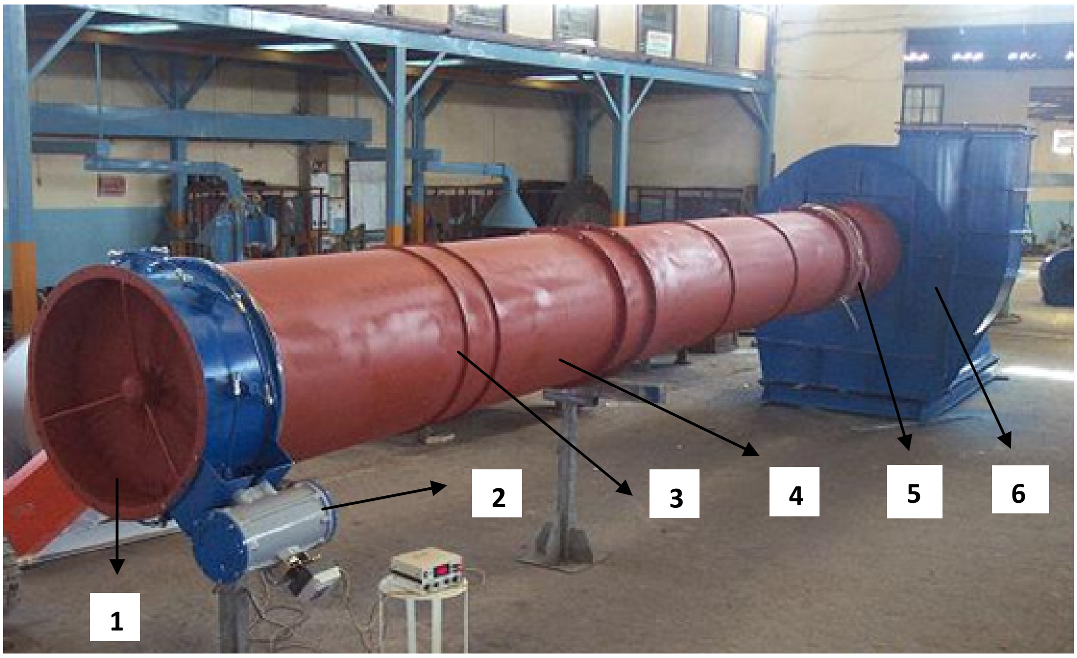
The terms σ_ε, σ_k , $c_{1\varepsilon}$ and $c_{2\varepsilon}$ in equation (11) are constants.

- In this research fan inlet velocity and outlet pressure of air were defined as boundary conditions. It was defined air volume created around rotor as rotatable and the remaining volume in the housing as solid and unmovable.

In order to obtain a fan characteristic curve, at least five pressure values corresponding to flow rate values at these five different points are necessary. Air velocity values measured at nine different points during the experiment were taken as the reference for the analyses performed on FLUENT. Here the aim was to get the possibility to compare the results with each other.

V. EXPERIMENTAL DETERMINATION OF THE CHARACTERISTIC CURVE OF THE DEVELOPED INDUSTRIAL CENTRIFUGAL FAN

Figure 13 exhibits the test stand and equipments used in the performance test of the developed industrial centrifugal fan. The centrifugal fan (6) whose characteristic curve was sought was fastened to the air suction line with a channel (4) having a diameter equal to that of the suction inlet and a length of at least 10D. An adjustable valve (1) is found at the other end of the channel open to the atmosphere. The position control of the valve is performed by a pneumatic actuator (2). In order to prevent the flow from being turbulent, a flow regulator (3) was placed at least 5D before the point where measurements were taken. A pitot traverse (5) was placed at the section where measurements were taken. Pitot traverse provides determination of the average air velocity at the section since it allows measurement of dynamic pressure at many points in the section. The properties and construction of the flow regulator and pitot traverse used in the test stand necessary for fan performance tests were defined by “Laboratory Methods of Testing Fans for Aerodynamic Performance Rating” present in AMCA 210-99 standard [7] (Fig. 13 -16).



No	Used Equipments	No	Used Equipments
1	Adjustable Valve	4	Channel
2	Pneumatic Actuator	5	Pitot Traverse
3	Flow Regulator	6	Centrifugal Fan

Figure 13. Test set-up of the centrifugal fan

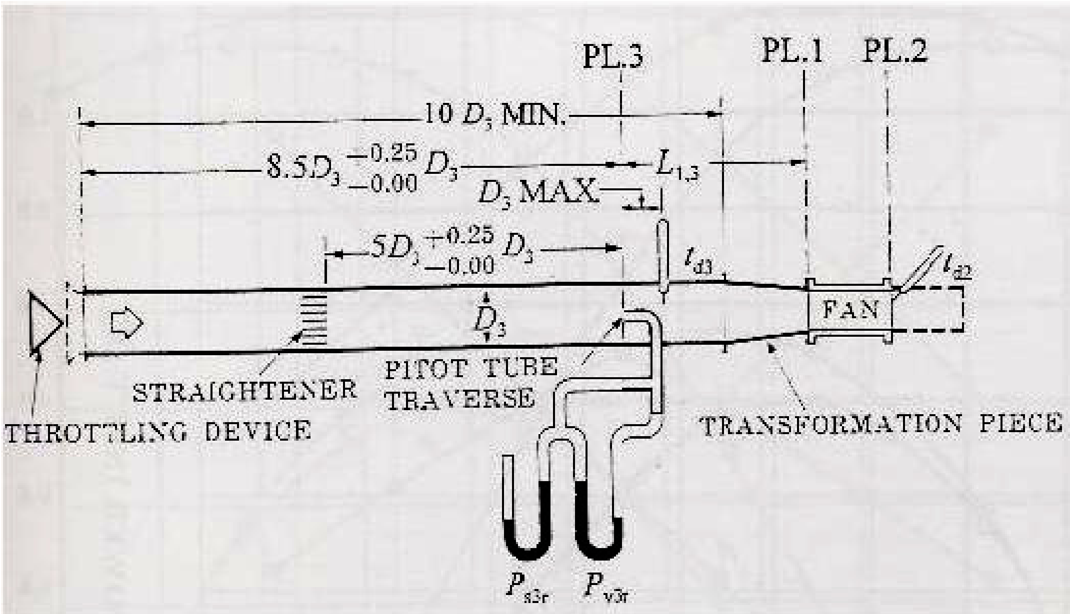


Figure 14. Test set-up of the centrifugal fan according to AMCA 210-99

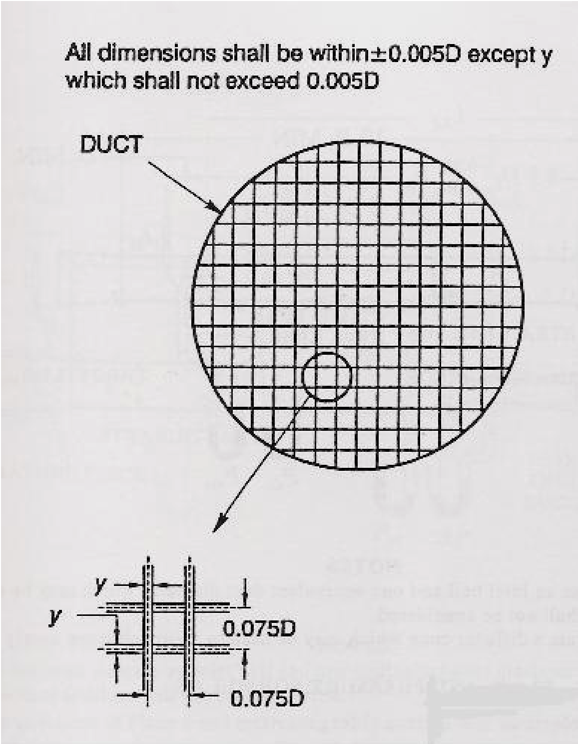


Figure 15. Flow straightener according to AMCA 210-99

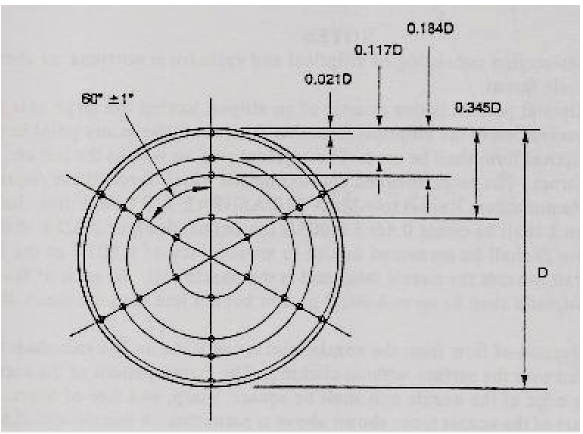


Figure 16. Pitot traverse according to AMCA 210-99

After the necessary test stand was constructed, all the test devices were calibrated. Five experimental test runs for each case were performed and the obtained results that were rather close to each others (0-3 %), were averaged. Measurements were taken by a pitot tube which is joined to the pitot traverse for different positions of the valve at the suction side of the channel in order to find out the characteristic curve of the fan experimentally. The valve could be adjusted by a pneumatic actuator from closed to wide open position. The measurements taken were the values of static and dynamic pressure, air velocity, current and voltage. The average air velocity and flow rate were calculated by means of dynamic pressure. Flow rate could also be calculated by taking the average of the air velocities measured at various points in the channel section by an anemometer. But it is more convenient to execute calculations using data taken from a correctly placed pitot traverse obeying the standards. The rotational speed of the fan was measured by means of a tachometer at each position of the valve at which the other measurements mentioned above also were taken. The form of the fan test was filled out with the values measured during the test. These values were reduced to the normal conditions (values of air at 20º C temperature at sea level) for considering the temperature and the altitude of the place where the test was performed (Table 2). The necessary calculations were performed by using equations (12-17) as follows:

$$p_T = p_{st} + p_{dyn} \tag{12}$$

$$u = \sqrt{\frac{2p_{dyn}}{\rho}} \quad [\mathrm{m/s}] \tag{13}$$

Here:

p_{dyn}: Dynamic pressure (Pa) measured by the pitot tube during the test (Table 2)

$$Q = 3600 A \cdot u \quad [\mathrm{m^3/h}] \tag{14}$$

Where:

A: Area of the duct cross section where measurements are taken (m^2) (Table 2)

u : Average air velocity in the duct (m/s) (Table 2)

Fan efficiency is calculated by means of equation (15):

$$\eta_f = \frac{Q \cdot p_T}{270000 \cdot N_{shaft}} \tag{15}$$

Where:

η_f = Fan efficiency (Table 2).

Q = Volumetric flow rate of the fan (m^3/h) (Table 2)

p_T = Total pressure delivered by the fan working at sea level at a temperature of 20 °C (mmH_2O) (Table 2)

N_{shaft} =Shaft power of the fan (HP) (Table 2)

$$N_{shaft} = N_m \ \eta_{mot} \ \eta_b \tag{16}$$

Here:

N_m = Power input from the mains (W), which is calculated by Eq. (17)

η_{mot} : Motor efficiency (90 %) to be taken from motor catalogue (Table 2).

η_b : Belt transmission efficiency to be taken as approximately 97 % (Table 2).

I_m = Current drawn from the mains (A)

$$N_m = \sqrt{3} U_m I_m \cos\varphi \tag{17}$$

I_m = Current drawn from the mains (A)

U_m = Grid voltage (V)

$\cos\varphi$ = Motor power factor to be taken from motor catalogue (see the variation of $\cos\varphi$ vs. load in Table 2).

Figure 17 gives the fan characteristic curve obtained as a result of the numerical analysis and the performance test carried out on the industrial centrifugal fan that was modeled and analyzed using FLUENT and manufactured at

Ed-Van Factory, where characteristic values of the fan were measured at 1310 RPM (Table 2).

VI. RESULTS AND CONCLUSION

The aim of this study was to carry out an industrial centrifugal fan's performance test by using the commercial software FLUENT and then to compare this with the performance obtained experimentally after manufacturing a prototype of the fan. According to the results obtained there is a maximum difference of 5 % between the values obtained numerically and experimentally. The maximum difference occurs at the beginning of the curve whose part is not significant in the selection of the fan. Therefore it may be concluded that the results obtained are in good agreement with the experiments. Figure 17 shows the comparison of the performance values obtained experimentally and numerically.

By examining the results it can be concluded that the trial-and-error method can be given up for new designs/redesigns of the fans and this process, namely changing the design or modeling a new design and performing numerical analyses on these till obtaining optimum results, can be executed using certain advanced software. But the research and development investment necessary for the computer hardware, software and personnel can be quite high. Nevertheless when high costs of the existing trial-and-error method as well as the business opportunities rising by new designs with reduced costs are considered, it can be concluded that this investment will amortize within a short time.

Table 2. Measured test values

	Velocity (m/s)	Chan. area (m^2)	Flow rate (m^3/h)	V^2 (m^2/s^2)	Dynamic pressure difference (mmH2O)	Measur. static pressure (mmH2O)	20°C Statik Basınç	20°C Total press. (mmH₂O)	U (volt)	I (A)	Cos φ	η_{mot}	η_b	1.73	0.00136	N_{shaft} (HP)	Efficiency (%)
1. Case	12.0	0.502	21686	144	2.8	248	242.7	242.7	395	38	0.85	0.9	0.97	1.73	0.00136	26.21	74.40
2. Case	15.0	0.502	27108	225	4.4	240	233.2	233.2	395	44.6	0.85	0.9	0.97	1.73	0.00136	30.76	76.14
3. Case	18.0	0.502	32530	324	4.3	230	223.4	223.4	395	47.3	0.85	0.9	0.97	1.73	0.00136	32.62	82.53
4. Case	20.0	0.502	36144	400	4.4	210	203.5	203.5	395	48	0.88	0.9	0.97	1.73	0.00136	34.27	79.51
5. Case	22.0	0.502	39758	484	7.0	188	179.2	179.2	395	49.3	0.88	0.9	0.97	1.73	0.00136	35.20	74.96
6. Case	23.0	0.502	41566	529	8.3	171	161.1	161.1	395	49.0	0.88	0.9	0.97	1.73	0.00136	34.98	70.88
7. Case	24.0	0.502	43373	576	9.0	153	142.6	142.6	395	49	0.88	0.9	0.97	1.73	0.00136	34.98	65.46
8. Case	25.0	0.502	45180	625	9.4	140	129.3	129.3	395	49	0.88	0.9	0.97	1.73	0.00136	34.98	61.84
9. Case	26.0	0.502	46987	676	12.5	127	113.4	113.4	395	49	0.88	0.9	0.97	1.73	0.00136	34.98	56.39

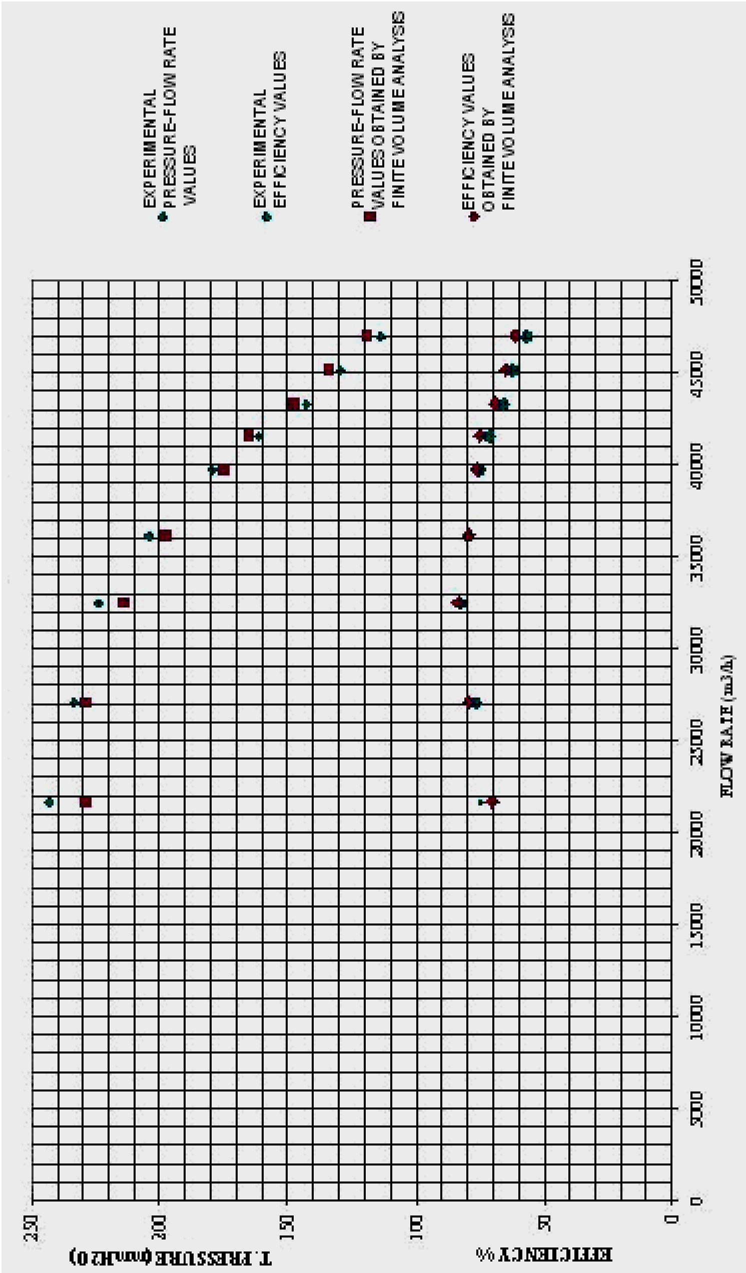


Figure 17. Comparison of the test results of the centrifugal fan with the values of performance curve obtained by finite volume method

VII. REFERENCES

1. Jeon, W.H., Lee, D.J., "A numerical study on the flow and sound fields of centrifugal impeller located near a wedge", *Journal of Sound and Vibration* 266 (2003) 785-804.

2. Lin, S., C., Huang, C.L., "An integrated experimental and numerical study of forward-curved centrifugal fan", *Experimental Thermal and Fluid Science* 26 (2002) 421-434.

3. Eck, B., "Ventilatoren: Entwurf und Betrieb der Radial-, Axial- und Querstromventilatoren", 4. Ed., Berlin, Springer, 1962.

4. Arun, N., Akkoc, H., "Fundamentals of Pneumatics" , TMMOB Publication Nr: 205, 1997.

5. Bohl, W. "Ventilatoren" , Vogel-Buchverlag, 1983.

6. Ugural, M., Parmaksızoglu, C., "Ventilators and Their Systems", Termas Publication, 1992. (In Turkish)

7. AMCA (Air Movement and Control Association) International, Inc. "AMCA 210-99 Laboratory Methods of Testing Fans for Aerodynamic Performance Rating", 1999

8. Fujii, K.L, Tamura, Y., "Capability of Current Supercomputers for the Computational Fluid Dynamics", Proceedings of the 1989 ACM IEEE Conference on *Super Computing*. PX7180, 1989, USA.

9. Engin, T., „Study of tip clearanceeffects in centrifugal fans with unshrouded impellers using computational fluid dynamics", Proc. IMechE Part A: J. Power and Energy, 220 (2006) 599-610.

10. Zhou, W., Zhao, Z., Lee, T. S., Winoto, S. H. Investigation of flow through centrifugal pump impeller using computational fluid dynamics. Int. J. Rotat. Mach., 2003, 9(1), 49–61.

11. Asuaje, M., Bakir, F., Kouidri, S., Rey, R. Inversedesign method for centrifugal impellers and comparison with numerical simulation tools. Int. J. Comput. Fluid Dyn., 2004, 18(2), 101–110.

12. Asuaje, M., Bakir, F., Kouhidri, S., Noguera, R., Rey, R. Computer-aided design and optimization of centrifugal pumps. Proc. IMechE, Part A: J. Power and Energy, 2005, 219, 187–193.

13. Yu, S. C. M., Ng, B. T. H., Chan, W. K., Chua, L. P. The flow patterns within the impeller passages of a centrifugal blood pump model. Med. Eng. Phys., 2000, 22, 381–293.

14. Yu, Z., Li, S., He, W., Wang, W., Huang, D., Zhu, Z. Numerical simulation of flowfield for a whole centrifugal fan

and analysis of effects of blade inlet angle and impeller gap. HVAC&R Res., 2005, 11(2), 263–283.

15. EN ISO 5801:2008, Part 1: Ventilatoren-Leistungsmessungen, Normkennlinien, 2008.

16. Bleir, F.P., "Fan Handbook", Mc Graw-Hill, 1978.

17. FLUENT, "Tutorial Guide", 2002