Numerical and Experimental Optimization of Flow Coefficient in Tubeaxial Fan

Alireza Falahat
Department of Mechanics, Mahshahr Branch, Islamic Azad University, Mahshahr, Iran
a_r_falahat@yahoo.com

Abstract— **In this study an attempt was made to find the best the best angle of attack and rotational velocity of a flat blade at a fixed hub to tip ratio for a maximum flow coefficient in an axial fan in a steady and turbulent conditions. In this study the blade angles are varied from 30 to 70 degrees and the rotational velocity is varied from 50 to 200 rad/sec for a number of blades from 2 to 6, at a fixed hub to tip ratio. The numerical and experimental results show that, the maximum flow coefficient is achieved at the blade angle of attack of between 45 to 55 degrees when the number of blades was equal to 4 at most rotational velocities. The numerical results show that as the rotational velocity increased, the flow coefficient increased but at very high rotational velocities the flow coefficient remained constant.**

Keywords— Angle of Attack, Tubeaxial Fan, Flow Coefficient and Flat Blade

I. INTRODUCTION

Axial fans are investigated widely because of their importance and their wide use in industry. In 1951, ECK [1] introduced many parameters on this issue and analyzed them as well. These types of work increased the efficiency of the axial fans. Theories on auto fan controls and section geometries on air channels was presented and studied by Wallis [2]. The explanations on these theories and also some work were done by Bleier [3], and finally he presented some formulas for pressure stagnation, available work and efficiency.

Fukano et al. [4] tried to measure a periodic velocity fluctuation downstream of the trailing edge of a rotating flat-plate blade using a hot-wire sensor. They showed the periodic velocity fluctuation due to karman vortex street downstream of the rotating flat-plate blade. Koop and Martinzzi [5] measured the mean velocity vector and derived the exit angle of an automotive engine cooling fan. They also found the mean flow angle is close to a local minimum of the probability density function of the swirl angle.

Sandra et al. [6] investigated how the changed of the blade pitch has a major influence on the unsteady operation of the axial flow fan. In their study, they measured the flow field at different radial locations on the inlet and exit planes. They revealed that the highest levels of unsteadiness are located at the hub and shroud regions.

In this research an axial fan in a shape of duct with a hub with a varied number of flat blades at one section of the fan is studied. The blade length was selected to be $21\,cm$, the chord of the blade is set to be $10\,cm$, the thickness of the blade $4\,mm$, the diameter of the hub $18\,cm$, the duct length as $70\,cm$, the duct diameter $62\,cm$, and therefore the distance between the tip of the blade and duct wall is equal to $1\,mm$.

To show numerical results validation, at a fixed number of blades, the mesh type and the number of nodes were changed so that the mass flow rate result obtained numerically was close to the result gained experimentally and with these numbers of nodes and mesh types, variables such as blade angle of attack and rotational speed were changed and numerical results were obtained. As the number of blades changed a different number of nodes were obtained.

The angle of attack (the angle between the flow and the blade), the number of blades and also the rotational velocity were varied and the mass flow rate was calculated. At first a set of 2 blades was placed on the shaft of the fan, and at one fixed rotational velocity the angle of attack was varied from 30 to 70 degrees. The mass flow rate was calculated for every angle of attack. Then the rotational velocity was changed from 50 to 200 rad/sec and the above calculation was repeated again. Finally the sets of blades were varied from 2 to 6 and the whole calculation was repeated for every set of blades (Fig. 1).

The aim is to find the best angle of attack, the best number of blades and the best rotational velocity for the maximum mass flow rate.

II. CFD MODELING

A. Boundary Conditions

In this paper, duct that rotate relatively is defined as moving wall. Moreover, as it is dependent to the fluid around it and as

It rotates. It is defined as relative to adjacent cell zone and rotational motion. Rotor is defined as stationary wall and inlet and outlet are defined as pressure inlet and pressure outlet (Fig. 2).

Fluid zone is defined as moving reference frame with rotational velocity in z-direction.

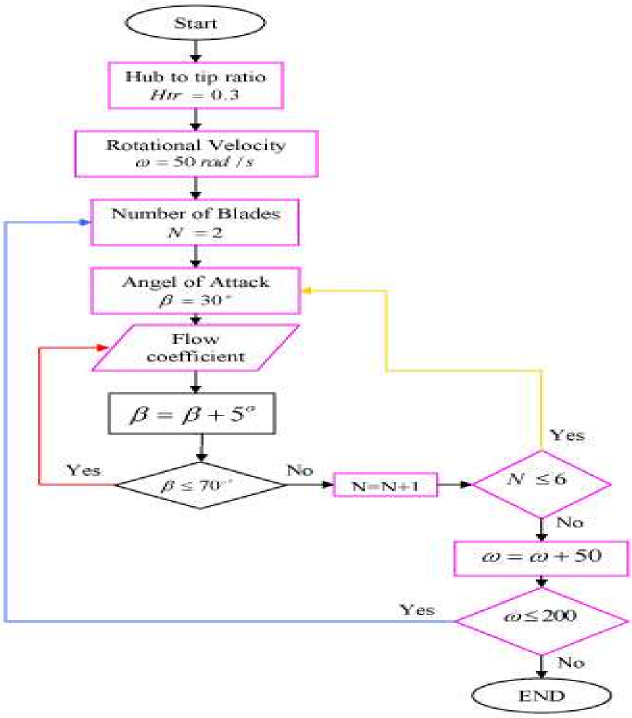

Fig. 1. Flow chart of analysis models

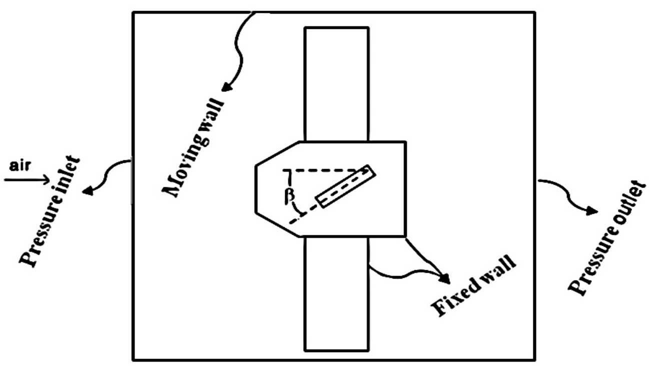

Fig. 2. Boundary conditions

B. Meshing

Fig. 3 shows an example of the mesh that is generated in our computation. At a fixed number of blades, the mesh type and the number of nodes were changed so that the mass flow rate data obtained numerically was close to the data gained experimentally and with these numbers of nodes the variables such as blade angle of attack and the rotational speed was changed and numerical results were obtained. The mesh type is unstructured T-gird consists of several nodes which differed from case to case. For example for case A shown in Fig. 4a, there were about 458662 tetrahedral cells, 29940 triangular wall faces, 888365 triangular interior faces, and about 91317 nodes. For case B shown in Fig. 4b, there were about 558923 tetrahedral cells, 47672 triangular wall faces, 1080021 triangular interior faces, and 112647 nodes. For case C shown in Fig. 4c, there were about 596795 tetrahedral cells, 54736 triangular wall face, 1152233 triangular interior faces, and about 120755 nodes.

The mesh points nearest to the body were much smaller that the ones further away. This approach of pure meshing or meshing technique caused a convergence to a numerical solution in less time steps when comparing some of the numerical results against the experimental results. Also this meshing technique could be the cause of a better estimation of the semi wake region with more accuracy. Numerical solution convergence was achieved in most cases at almost 500 time steps.

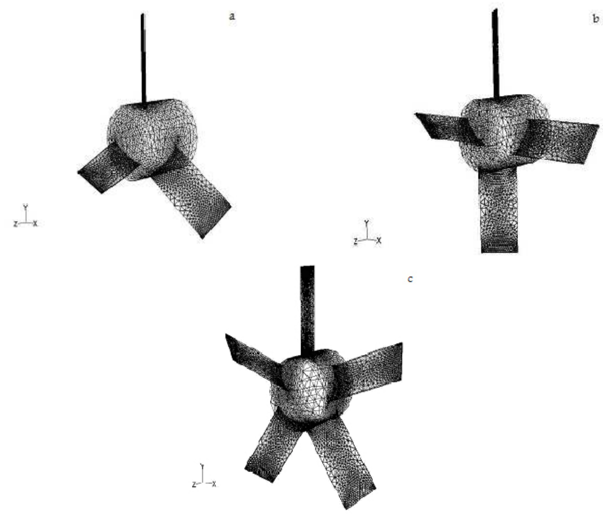

Fig. 3. Mesh points on the hub and also on the fan blades, a) three blades, b) four blades, c) five blades

C. Numerical Method

Let us consider a specific three dimensional and temporary domain. The spatial and temporary coordinates are denoted by:

$\vec{y} = (x, y, z)$ And $t \in (0, T)$

The Navier-Stokes equations of 3D flows:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{V} \right) = 0 \quad (1)$$

$$\frac{\partial (\rho u)}{\partial t} + \nabla \cdot \left(\rho u \vec{V} \right) = -\frac{\partial p}{\partial x} + \rho f_x + (\Im_x)_{vis} \quad (2\text{-a})$$

$$\frac{\partial (\rho v)}{\partial t} + \nabla \cdot \left(\rho v \vec{V} \right) = -\frac{\partial p}{\partial y} + \rho f_y + (\Im_y)_{vis} \quad (2\text{-b})$$

$$\frac{\partial (\rho w)}{\partial t} + \nabla \cdot \left(\rho w \vec{V} \right) = -\frac{\partial p}{\partial x} + \rho f_z + (\Im_z)_{vis} \quad (2\text{-c})$$

Here ρ, u, v, w, v, p, f , and $\Im$ are the density, velocity component in the x, y, z direction, velocity, pressure, external body forces, and viscous forces respectively. For the problem under consideration, the fluid was assumed Newtonian, and the flow is taken as turbulent. When turbulent, the dynamics viscosity of flow is modified locally using a k-epsilon turbulent model. No slip boundary condition is exerted on the wall; also the normal velocity on external surfaces of the wall is forced and set to zero. On the wall:

$$\vec{V}.n = 0 \tag{3}$$

$$\frac{\partial \phi}{\partial n} = 0 \tag{4}$$

The axial flow velocity, V_a, in this design is needed which can be computed from the following equation:

$$V_a = \frac{Q}{\frac{\pi}{4}(D_{fan}^2 - D_{hub}^2)} \tag{5}$$

In the above equation, D is diameter and Q is flow rate.

The fluid flow coefficient, ϕ, can be calculated from the following equation:

$$\phi = \frac{V_a}{\omega \frac{2\pi}{60} \times \frac{D}{2}} \tag{6}$$

In the above equation, ω is rotational velocity.

III. EXPERIMENTAL ANALYSIS

Fig. 4 shows the test axial fan, the model consisted of a duct with a four-blades fan that the blades attached to hub with bolt. Beseline model parameters are shown in Table 1. Angle of attack varied from 30 to 70 degrees. In the every angle of attack, velocity is measured in ten point of fan exit (Fig. 5) with anemometer. As such testing conditions where the temperature is $17^o C$, static pressure is $88872.7\,Pa$ and density is $1.1\,kg/m^3$.

It should be noted that number of point according to distance from the fan axis is chosen.

TABLE 1: MODEL PARAMETERS

Duct	Inner diameter	62 cm
	Duct length	70 cm
Fan	No. of blades	4
	Diameter	60 cm
	Hub to tip ratio	30%
	Tip clearance	1 cm
	Angle of attack	30^o to 70^o
	Rotational speed	200 rad/Sec
	Blade thickness	2 mm
	Power of motor	3 hp

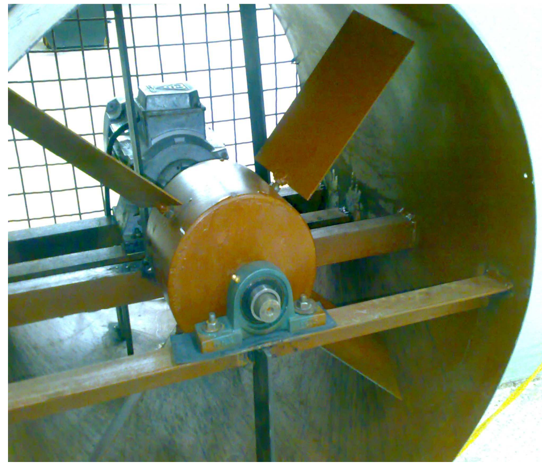

Fig. 4. The test axial fan

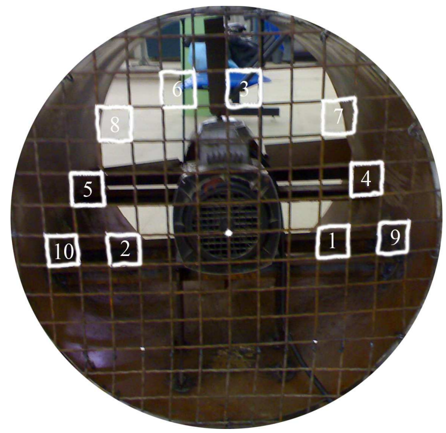

Fig. 5. Points of measuring in fan exit

IV. RESULT AND DISCUSSION

Fig. 6, Fig. 7 and Fig. 8 show the numerical solution of flow coefficient variation as the blade angle of attack in different rotational velocities. In these Figures can be seen that with increasing angle of attack from 30 to 50 degrees, flow coefficient increased and then with increasing angle of attack from 50 to 70 degrees, flow coefficient decreased.

In Fig. 9, results for four-blade condition and 30 percent of hub to tip ratio in a constant rotational velocity are compared with a real constructed model. It is observed that the optimum angles of attack obtained from the experiment results are in good agreement with the numerical results. Although, some differences observed at some points, but these can be interpreted as faults in measurements and experimental model construction.

With increasing number of blades (Fig. 10), flow coefficient increases until four blades in every blade angle of attack then variations from four to six blades are very low.

Fig. 11a and Fig. 12a, show the flow velocity vector and pressure pattern around the blades at 30-degree angle of attack. Comparing Fig. 11a, 11b and 11c, we can say that at very low (30 degree) and at very high (70 degree) angles of attack, we see a larger wake behind the blades at the tips (Fig. 11a and Fig. 11c) and a more smooth flow pattern around the blades when the blade angle of attack was about 50 degrees (Fig. 11b). The pressure distribution in Figures 12a, 12b and 12c show the same thing.

Fig. 13 shows the velocity magnitude on the blade from the root to the tip at two rotational velocities. We can see from these two figures that at higher rotational velocities ($200\ rad\ /\ Sec$), a larger velocity magnitude (about $15\ m/s$) exists around the blades.

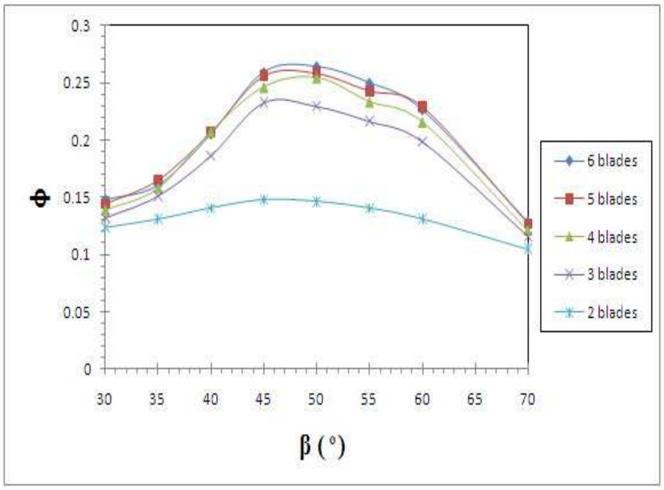

Fig. 6. Flow coefficient variation with respect to blades angle of attack at $100\ rad\ /\ Sec$ for a fan with several number of blades

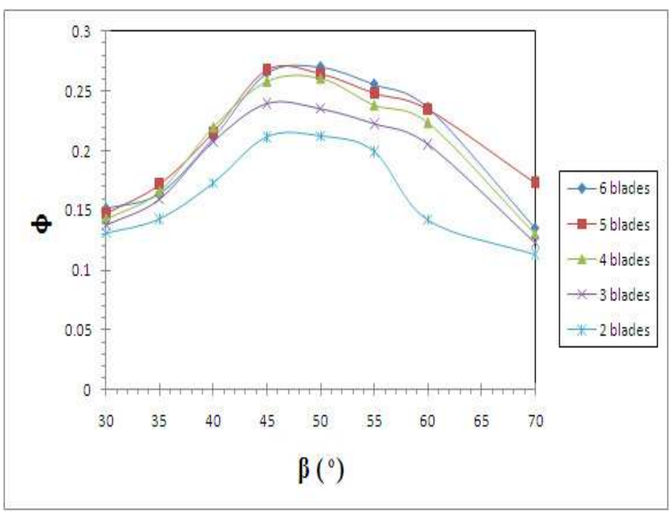

Fig. 7. Flow coefficient variation with respect to blades angle of attack at $200\ rad\ /\ Sec$ for a fan with several number of blades

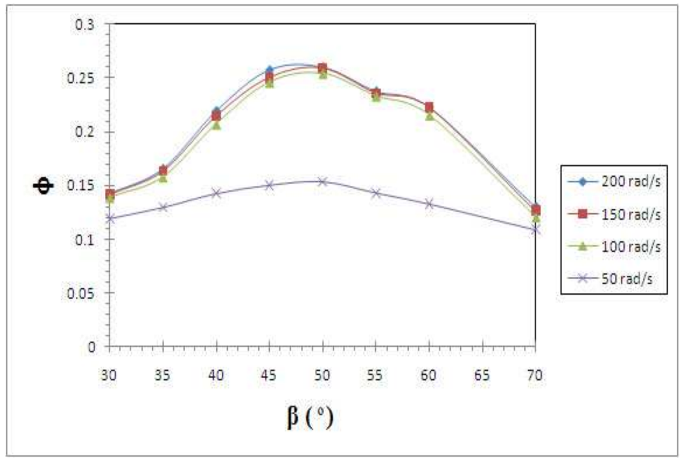

Fig. 8. Flow coefficient variation with respect to blades angle of attack at $100\ rad\ /\ Sec$ for a fan with four blades

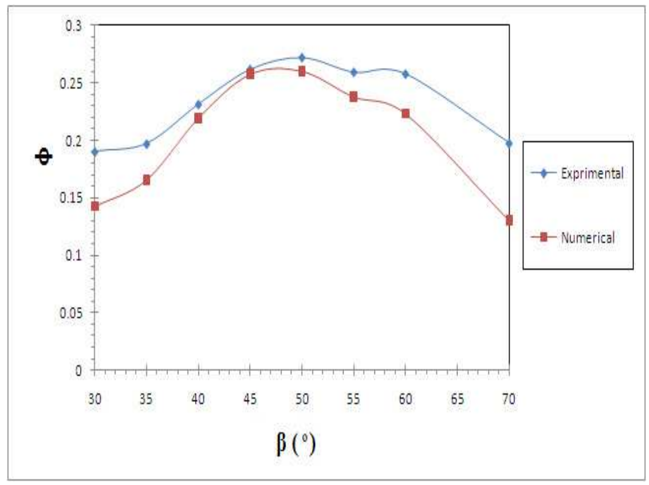

Fig. 9. Comparing experimental and numerical flow coefficient of four blades fan with respect to blades angle of attack

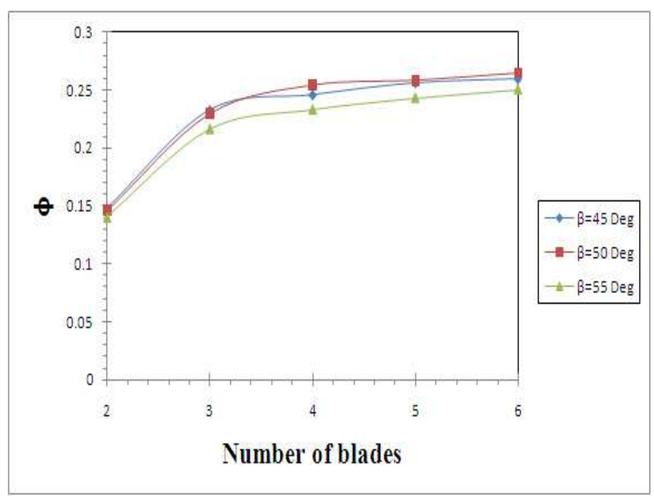

Fig. 10. Flow coefficient variation with respect to number of blades in optimum angle of attack and $100\ rad\ /\ Sec$

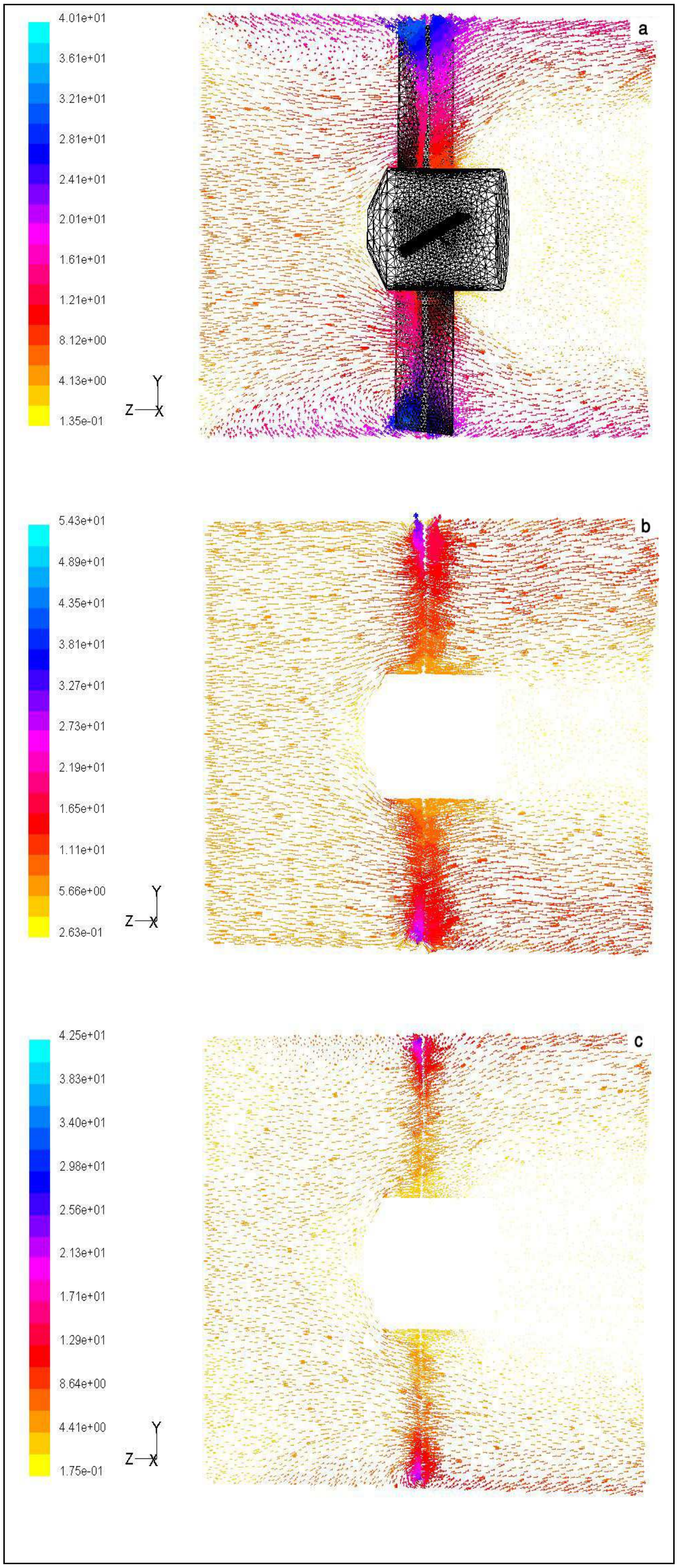

Fig. 11. Velocity vectors in axial direction of a fan with four blades at $100\ rad\ /\ Sec$ for three blade angles of attack: a) 30 degree, b) 50 degree and c) 70 degree

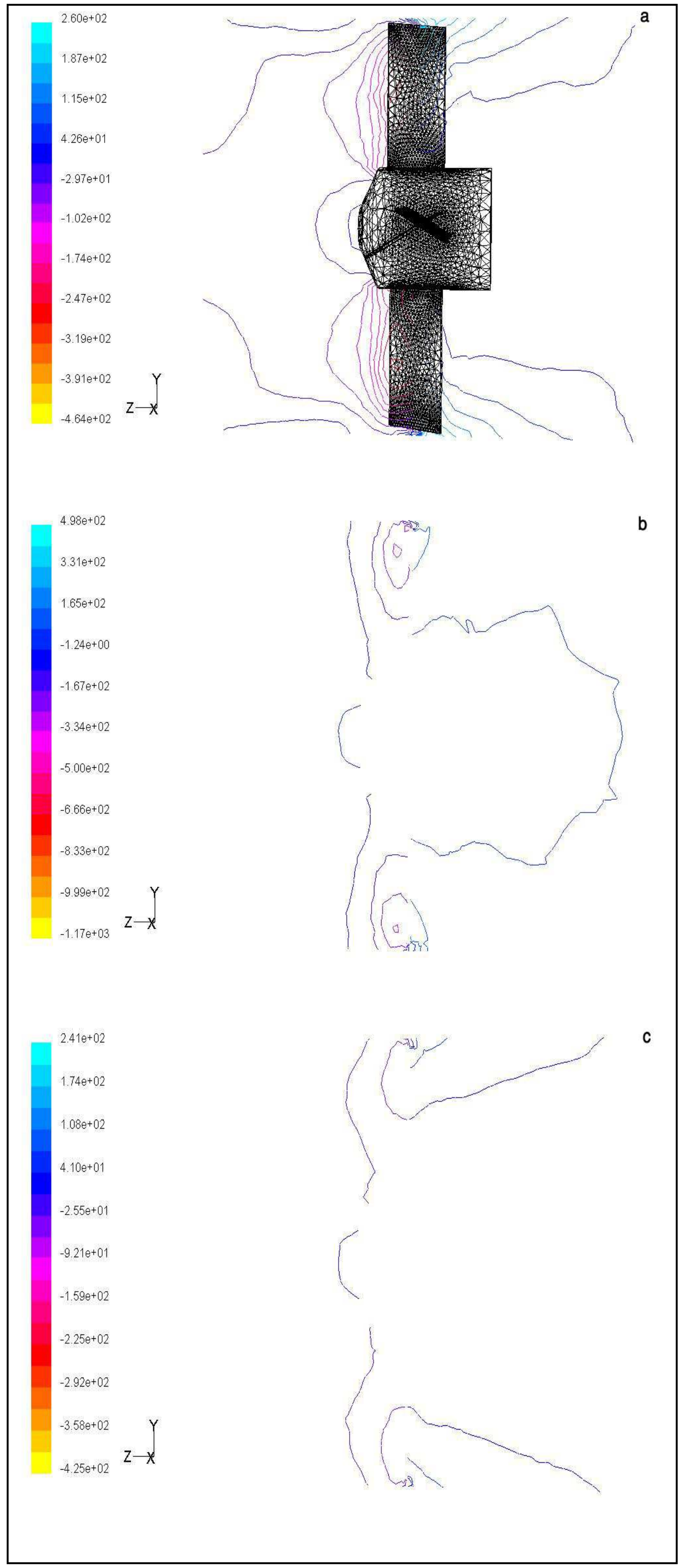

Fig. 11. Pressure contours of flow in axial direction of a fan with four blades at $100\ rad\ /\ Sec$ for three blade angles of attack: a) 30 degree, b) 50 degree and c) 70 degree

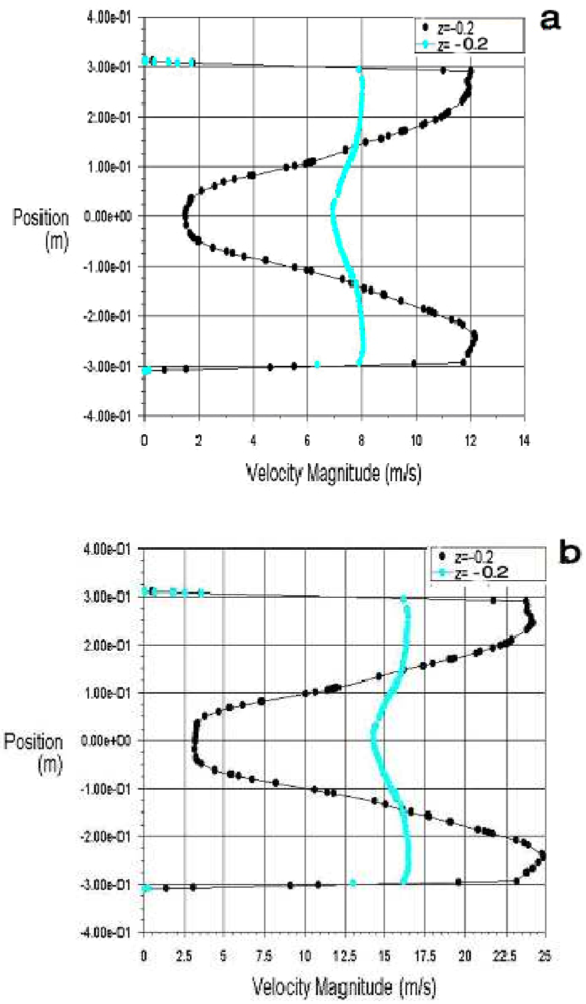

Fig. 12. Velocity magnitude on both sides of a blade of a fan with four blades at 50 degree angle of attack in two revolutions

V. CONCLUSION

The present study shows the effect of the blade angle of attack, the number of blades and also the rotational velocity on the flow coefficient in an axial fan with flat blades.

It can be seen from Fig. 6 that, fan with two blades has the minimum flow coefficient at all blade angles of attack. We see that at a fixed blade angle of attack, as the number of blades increases, the flow coefficient increases also, but the flow coefficient did not changed very much for a fan with four, five or six blades.

From Fig. 6 and Fig. 7 we can see that at about 45 to 55 degrees angle of attack the flow coefficient was at maximum. From Fig. 8 we can see that as the rotational velocity increased from 100 to 200 rad/Sec the flow coefficient was almost doubled.

We can conclude that in an axial fan with flat blades, at every rotational velocity a maximum flow coefficient can be achieved when the fan has 4 blades with 45 to 55 degrees blade angles of attack. When the rotational velocity increases, the flow coefficient increases also.

REFERENCES

[1] B. ECK, “Fans: Design and Opration of Centrifugal, Axial-Flow and Cross-Flow Fans (Translated and Edites by Azad, R. S, and Scott, D. R),” Pergamon, Oxford, 1973.

[2] R. A. Wallis , “Axial Flow Fans and Ducts,” Wiley, New York, 1983.

[3] F. P. Bleier, “Fan Handbook,” *McGraw-Hill, New York, pp. 13-117.*

[4] T. Fukano, H. Saruwatari, H. Hayashi, H. Isobe, and M. Fukuhara, “Periodic Velocity Fluctuation in the Near Wake of Rotating Flat-Plate Blade and Their Role in the Generation of Broadhand Nois ,” J. Sound Vib., 181, pp.53-70, 1995.

[5] G. A. Koop, R. J. Martinuzzi, “The Measuriment of Mean Flow Angles Between an Automotive Fan and Stator,”Proceeding of the 2002 ASME Fluid Engineering Division, ASME, Canada, FEDSM2002-31431, 2002.

[6] V. S. Sandra, B. T. Rafael, S. M. Carlos, and B. M.Eduardo,“Total Unsteadinees Downstream of an Axial Flow Fan with Variable Pitch Blades,” ASME: J. Fluid Fng., 124, pp. 280-283, 2002.

Alireza Falahat
Faculty Member of Department of Mechanical Engineering, Mahshahr Branch, Islamic Azad University, Mahshahr, Iran.
E-mail: a_r_falahat@yahoo.com
a.falahat@mahshahriau.ac.ir
Tel: +98 652 2358994