NUMERICAL SIMULATON OF AXIAL FLOW FAN USING GAMBIT AND FLUENT

Srinivas G[1], Srinivasa Rao Potti[2]

[1]*Asst.Professor, Dept of Aero & Auto Engg, MIT-Manipal University,Manipal, Karnataka-576104*
[2]*Assoc.Professor,Dept of Hum & Mngt, MIT-Manipal University,Manipal, Karnataka-576104.*

Abstract

The main objective of this paper consists of the computation of the aerodynamic performances of symmetrical blade contours of fully reversible axial fans by Numerical (CFD) methods, developing by a public literature survey, methodology for the design of reversible axial fans and analysis of the designed fan with CFD methods. The aerodynamic shows of the blade cascades are evaluated using FLUENT 6.0 software for different boundary conditions, strengths and angles called Angle of Attack(AoA) cascade also performing for varying speeds. The consequences of these computations are embedded into the industrialized methodology. Which is designed with the developed methodology, is done with Numerical methods.

Keywords—*fan; numerical; CFD; axial; design*

---***---

1. INTRODUCTION

A power-driven fan is a device used to create flow within a fluid, naturally a gas such as air. The fan contains of a rotating prearrangement of vanes or blades which act on the air[1,2,4]. The rotating muster of blades and hub is recognized as an impeller, a rotor, or a runner. Generally, it is contained within some form of housing or case. This may direct the airflow or upsurge safety by averting objects from contacting the fan blades[3,8].

Fig 1: Typical Power driven box Fan

Maximum fans are powered by electric motors see Fig 1, but other sources of power may be used, counting hydraulic motors and internal combustion engines.

Fans create air flows with high volume and little pressure (although higher than ambient pressure), as opposed to compressors which produce great pressures at a moderately low volume[7]. A fan blade will often revolve when exposed to an air stream, and devices that take benefit of this, such as anemometers and wind turbines, frequently have designs similar to that of a fan. Therefore, fans may become ineffectual at cooling the body if the nearby air is near body temperature and contains high moisture[5,6,8].

2. NUMERICAL TECHNIQUES (CFD)

The transport of fluid comprises gases/liquid from one component to other in power/process equipment are described through mass, momentum and energy conservation principles. The Navier Stokes (transport) equations are derived from these principles and are discussed by Hoffman, K.A [1993] which are represented mathematically as-

The terms on Left Hand Side (LHS) defines acceleration of flow over time with inertia depends on the sum of the external forces, diffusion and sources acting on the fluid element. If the value of φ is 1, the eqn. (1) results in continuity equation. If the value of φ is either u or v or w, the above eqn. describes momentum equation in x, y, z directions. If the value of φ is h then the above eqn. yields to energy equation.

$$\frac{\partial \rho \varphi}{\partial t} + div(\rho \varphi \vec{u}) = div(\Gamma grad\varphi) + q_\varphi \qquad (1)$$

In order to resolve wide spectrum of scales in turbulent eddies, normally two approaches are employed. This requires dense mesh points for proper resolution and its solution depends on heavy computational resources that are expensive, time consuming process and therefore very rarely used simulation technique. The other approach generally used for most of the applications are Reynolds' averaging process wherein flow variables are decomposed into mean and fluctuating components as

$$u_i = \overline{u}_i + u_i^{'} \quad (2)$$

Where i=1.2,3 denotes in x, y, z direction [7]. Likewise the pressure and other scalars can be expressed as

$$\phi = \overline{\varphi} + \phi^{'} \quad (3)$$

Substituting flow variables in this form into the instantaneous continuity and momentum equations and taking a time (or ensemble) average (and dropping the over bar on the mean velocity) yields to

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i\right) = 0 \quad (4)$$

Eqn (4-

$$\frac{\partial}{\partial t}\left(\rho u_i\right) + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{i,j}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(\overline{-\rho u_i^{'} u_j^{'}}\right) \quad (5)$$

5) are called RANS equations. The term $\overline{\rho u_i^{'} u_j^{'}}$ in the eqn (4.5) results from averaging process and is called Reynolds' Stress. With the help of Boussinesq hypothesis to relate the Reynolds stresses, choosing Kronecker delta δ=1 if *i=j* and $\overline{u_i^{'} u_i^{'}} = 2k$ the Reynolds's stress term in the eqn (5) is rewritten as –

$$-\rho \overline{u_i^{'} u_i^{'}} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t \frac{\partial u_k}{\partial x_k}\right)\delta_{i,j} \quad (6)$$

Where μ_t is turbulent viscosity. To resolve turbulence viscosity and Reynolds' stresses, eddy viscosity models based on Boussinesq hypothesis will leads to zero, one and two equation turbulence models and Reynolds's Stress Models (RSM). The strength and weakness of these models for prediction of turbulence effects are extensively studied.

3. METHODOLOGY

The data has been captured from the public literature survey and can generate the geometry and grid by GAMBIT software (see Fig 2). After a grid has been read into FLUENT, all remaining operations are done within its solver. These contain setting boundary conditions, defining fluid properties, performing the solution, refining the grid, and viewing and post processing the results.

For the successful design of any fan blade we need minimum performance parameters.

Table- 1: Axial flow fan Solidity at blade regions

Solidity Value	Blade Hub	Blade Mean	Blade Tip
0.1	0.22	0.41	0.2
0.2	0.47	0.41	0.12
0.5	0.65	0.41	0.09
1	0.65	0.29	0.05
1.5	0.6	0.23	0.02
2	0.56	0.28	0.01

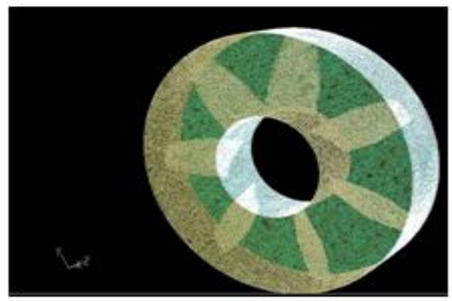

Fig-2: Numerical Meshing around the Fan

Table-2: Axial Flow fan boundary conditions

Fan Boundary	Conditions
Inlet	Pressure Inlet
Outlet	Pressure outlet
Wall Boundary lower	Hub
Wall Boundary Upper	Casing
Pressure side	Periodic
Suction Side	Periodic
Inlet Zone	Pressure inlet
Zone outlet	value -119 Guage pressure
Fluid	air
Fluid Speed	400 rpm
Turbulence	RNG model
Near wall treatment	Standard Wall Functions
Type of model	Viscous
Iterations for convergence	1.00E-05
Temperature	298K
Operating pressure	430Pa

So in this paper even we calculated the example mean diameter design with a solidity of σ =0.2 is existing mathematically to have a better understanding of the design procedure. Note that following design is made through the theoretical standard calculations shown in Table 1.

The Numerical mesh of the result domain involves of tetrahedral elements and triangular prisms. Because the domain replications itself in every blade, the entire domain is not modeled for the required solution, only the size around one blade is meshed shown in Fig 3.

The number of cell elements in the domain is approximately 150000 and number of nodes is 30000, which balance sheet for one seventh of the entire fan. Figure 2 shows the Numerical tool mesh of the solution. In the figure, the inner elements are not shown to have the full perceptibility of the blade. Figure 2 is the mesh of the entire fan, which is built by seven periodic replications for seven blades.

The boundary settings of the result domain resemble those in the two dimensional cascade studies. Like the boundary conditions of the two-dimensional blade cascade results, the inlet and outlet surfaces of the mesh are Flow Inlet/Outlet type limitations and the side faces are periodic boundaries. The difference is that the inlet face is defined as pressure inlet boundary, instead of velocity inlet boundary. Also, in three-dimensional fan analysis, the intermittent boundary conditions are rotationally periodic, while the periodic boundaries in two-dimensional cascade analysis are transnationally intermittent.

4. RESULTS AND DISCUSSION

The consequences are obtained with the solution of the continuity and N-S equations along with the equations for the designated turbulence model. For the solution of the turbulence, RNG k-ε model is carefully chosen.

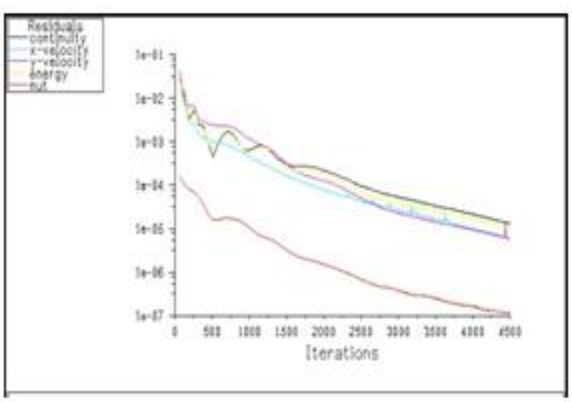

Fig-3: Iterations convergence plot

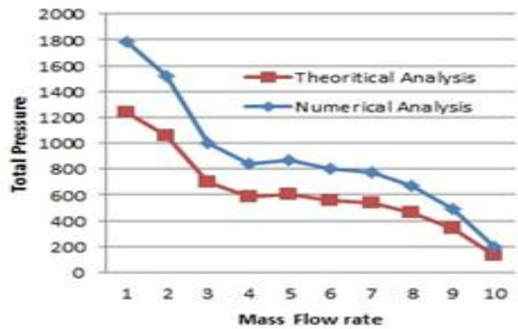

Fig-4: Analysis result validation

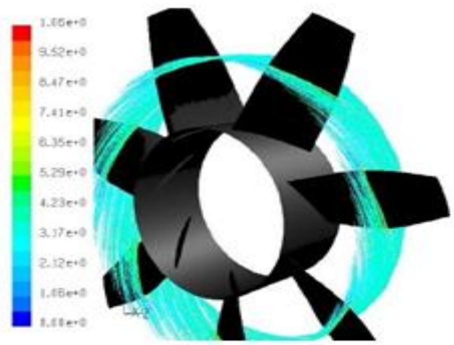

Fig-5: Pressure intensity on axial flow fan

The iterations for the performance points are made by allocating diverse gauge pressure values at the exit boundary of the domain, resulting in consistent flow rates for the pressure gaps across the fan. The inlet pressures are kept constant but the outlet pressures are altered. At the end the total solution and the performance curve for the fan are given. The iterations solution convergence plot shown in Fig 3.

The results of the computations give a total pressure rise of 438 Pa across the fan for the design flow rate of 8m3/s. This 6 % difference between the design pressure rise and numerical analysis can be interpreted as close and the results are in good agreement shown in Fig 4.

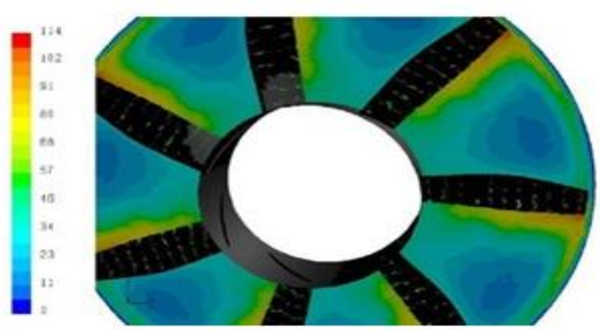

Fig-6: Temperature distribution on Fan

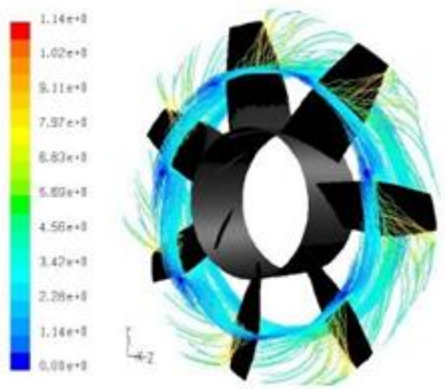

Fig-7: Velocity streamlines on Fan

After successful completion of the analysis of the axial flow fan it can be see that the path lines are suave at the mid-span and at the hub; there is no split-up of the flow either from the blades and the hub. at the lowest point of the delaying dip , the path lines crossways the rotor start to separate from the hub surfaces of the fan while the path lines show a favorable trend at the mid-span for the same operating point. This separation is caused by the separation of the stream on the back of the blades due to improved rates. The elements separated from the hub are collected on the front side of the next blade, so a vortex Structure is formed at the tip of the blade shown from Fig 5 to 9.

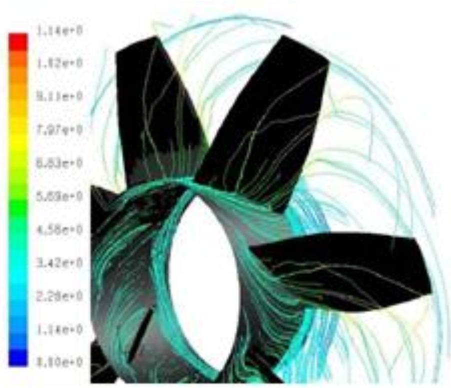

Fig-8: Velocity streamlines on Fan at tip region

Since this point to shut off, the flow passes the rotor no longer axially but at a slope. As the shut off is advanced, the vortex at the tip of the blade gets larger and the separated stream from the hub extends into the blade passage. The flow permits the rotor more or less centrifugally and therefore the pressure rises as in the radial blades. The outlines of velocity magnitudes for the operating point, the velocity colors in Figure has usual tendency for axial turbo machinery while the damage in the velocity magnitude at operating point is observed inside the blade channel . The performance of the calculated fan is examined thoroughly. The performance points and the fan curve are obtained. Also, a deep view of the fluid flow inside the blade passages and across the fan is explored.

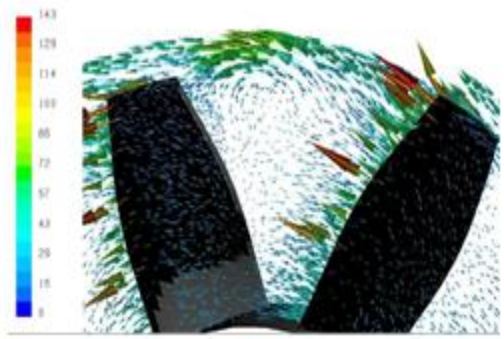

Fig 9: Turbulence intensity on Fan blade

The analysis results for the comparison analysis of the data obtained for the design rotational speed and the simulation results for the augmented speed are shown in Fig 4 . The data on Table 1 are expressed by solidity the fan performance blade curves from hub to tip shown in Fig 8 for the design speed and increased rotational velocity streamlines. The better rotational speed curves include both the similarity analysis and simulation consequences.

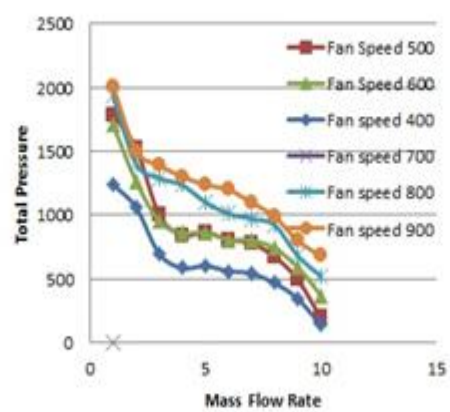

Fig 10: Axial Fan Total pressure for varying Speeds

The curves for the increased rotational speed show almost the same trend in the recommended operation range. After the fan stalling, little discrepancy is observed. The results of the simulation for this region can be interpreted as satisfactory because the flow in this region of the performance curve is quite chaotic inside the blade passages thus the analysis may not predict consistent results with the comparison analysis. Briefly, the results in the recommended operation range are quite well and the results in the region left to fan stalling point are in acceptable series for varying speeds shown in Fig 10.

5. CONCLUSIONS

The aerodynamic performances of the Axial flow fan blade profiles for bidirectional operation are acquired from numerical simulation via FLUENT and GAMBIT. The separation of the flow over the blades takes place for these increased incidences, sense that the fan is operating in the stalling range. Besides, the fan was also operated at an increased rotational speed of 900 rpm. The Numerical simulation results are surrounded into the developed theoretical procedure. The aerodynamic performance of the designed fan is simulated for the different design rotational speed and an increased rotational speed, results of which are compared with the simulated analysis. The incidences of the blades are also altered as well as the solidity. The coefficients of the profiles are obtained for arrangements of the above variables, air stream velocity, solidity and angle of attack (AoA). The most significant consequence of this phenomenon is the increase of the incidence angle of the blades that are set for the design range. When the solutions are compared with the similitude analysis, it is observed that the simulations yield results that are close enough to those of similitude analysis in the operation range but little discrepancy is detected in the stalling range due to highly slanted nature of the flow in this area but still it can be concluded that the solutions are stable. Finally the theoretical results and Numerical results of axial flow fan are suitably matching with the Minimum errors for good aeromechanical features.

REFERENCES

[1]. Xu, C., "Numerical Study of Flow and Heat Transfer in Turbomachinery" Ph.D. Dissertation, The University of Wisconsin-Milwaukee, December 2000.
[2]. Tulapurkara, E.G., "Turbulence Models for the Computation of Flow Past Airplanes", Prog. Aerospace Sci. Vol. 33, 1997, pp 71-165.
[3]. Jaw, S.Y., Chen, C.J., "Present Status of Second Order Closure Turbulence Models", Journal of Engineering Mechanics, May 1998, pp 485-501.
[4]. Wilcox, D.C., "Turbulence Modeling for CFD", DCW Industries, 1998.
[5]. Ferziger, J.H., Peric M., "Computational Methods for Fluid Dynamics", Springer-Verlag, 1996.
[6]. Fluent 6.0 Documentation, FLUENT INC.
[7]. Wallis, R.A., "Axial Flow Fans and Ducts", John Wiley and Sons, 1983.
[8]. Saravanamuttoo, H.I.H., Rogers, G.F.C., Cohen, H., "Gas Turbine Theory", Prentice Hall, 2001.
[9]. Wilson, D.G., Korakianitis, T., "The Design of High Efficiency Turbomachinery and Gas Turbines", Prentice Hall, 1998.