Design and Analysis of an Axial Fan applicable for Kiln Shell Cooling

Parasaram Sarath Chandra[1], Dr. K. Sivaji Babu[2], U. Koteswara Rao[3]
M.Tech (Machine Design) Research student[1], PVPSIT, Kanuru, India.
Principal[2], Department of Mechanical Engineering, PVPSIT, Kanuru, India.
Associate Professor[3], Department of Mechanical Engineering, PVPSIT, Kanuru, India.

***Abstract* - Present paper deals with design and analysis of an axial fan applicable in kiln shell cooling. The design of various elements of axial fan is discussed in detail. Design validation was done by comparison with CFD analysis. A relation between volumetric flow and pressure is established with graphical approach which is a performance analysis. The main scope of the design process of an axial fans, is to deliver high efficiency blades. Several aerofoils have been studied for the application and finally a combination of airfoil literally called F-series had been used in the present work. Efficiency has been increased when compared with the existing machine from 52% to 63% of total to total efficiency. Performance of the new design has been verified with commercial CFD software, ANSYS CFX.**

***Keywords* - Axial fans, airfoils, free vortex theory, performance analysis, CFD.**

I. INTRODUCTION

The main focus of the paper was to design and analyse an axial fan with improvised efficiency, by using an industrial example for demonstration purpose and establish a systematical design procedure which predicts the fan performance where usage of computational fluid dynamics (CFD) will be an instrumental in practice. High volume flow and low pressure fans are used in cooling applications for several process equipment, automotive, and also for ventilation purposes. Present paper focuses on a specific application related to clinker kiln-shell cooling.

A comprehensive aerodynamic treatment of ducted axial flow fans has been presented in the present work. Airfoil and the blade so designed are analysed using CFD software. The main emphasis will be on improvement of efficiency and the system performance. Fans are a kind of equipment where we can use engineering strategies and optimize the energy consumption without effecting their efficiency. They are mainly being used for ventilation and cooling purposes at industrial plants. To cool a kiln-shell used in cement plant needs a fan with specific speeds which works for given site conditions. Whether it may be a blower or fan which is classified as an industrial fan, selection of a fan and design of impeller needs a keen study of rotor blade design which is majorly based on velocity components. The blade may be of simply a plate with camber angles or an aerofoil shape. Research is suggesting that replacing the curved camber plate with the aerofoil blade may produce almost identical performance but which results in the considerable increase in the structural strength of the blade.

An axial flow fan can achieve high efficiencies as with an optimum blade settings and is only slightly lower than that obtained with the backward inclined aerofoil centrifugal fan [14].

Present work deals with axial flow of type power absorbing turbomachines. The flow in the investigated form, i.e., air, is characterised by Mach numbers below the compressibility limit. It is a clear case that fan operating with incompressible flow, which is a type of high capacity, low head (pressure), and single stage axial flow type turbomachine.

Designing an efficient airfoil profile implies that the shape which thus acquired has to be aerodynamically efficient.

Fan characteristics can be described by consistent parameters such as volume flow rate, pressure, power, and efficiency.

An industrial example, upon industrial study, has been considered for the demonstration of the design flow. Based on specifications of site conditions and flow requirements, a preliminary calculations has been made which defined the system resistance. From these considerations a specific valve of fan diameter has obtained, by using Cordier diagram which was verified with background fan curves for an optimal value.

II. DESIGN APPROACH

A. FAN THEORY

Based upon initial calculation of pressures, velocity of air, speed of an impeller within the system resistance with trial and error method will give non-dimentionalized factors like load factor, specific speed and specific diameter.

The design of high efficient fans is often based on the experience of the designer. In order to determine the dimensions of a fan, one can use either Cordier diagram or background curves. Cordier diagram provides an optimum dimensions for a fan as a system, whereas through background curves, system and blade geometries could be approximately estimated and refined for actual design point based on well-established design procedure. The present paper sees, the use of both in a blend for optimum values and to attain best efficiency point.

Direction of flow is not a concern in the regular maintenance, but sometimes it may affect the magnitude of system effect factors. So as a code of practice, manufacturers prefer to use ISO 13349 standard, which specifies that the rotation is determined from the side opposite the inlet (as shown in the below figure 1)

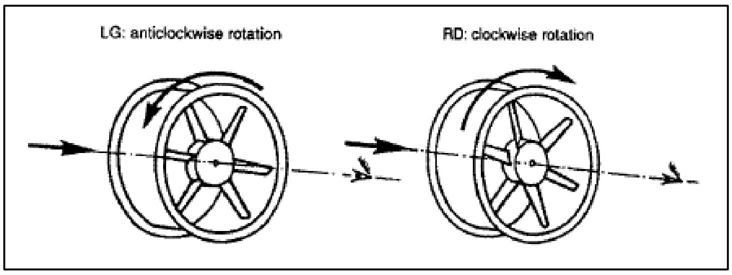


Fig. 1 Direction of rotation of axial and mixed flow fans [2]

The operating principle of an axial flow fan is simply a deflection of air from suction side to pressure side of an aerofoil section. An axial fan relies on the same principle as an aircraft propeller, although usually with many more blades for mine and industrial applications. Highly recommended to be used for high volume flow rate, low pressure, low or medium speed and non-ducted systems.

Air passes through the fan along flow paths that are essentially aligned with the axis of rotation of the impeller and without changing their macro-direction. However, later in the chapter we shall see that significant vortex action may be imparted to the air. The particular characteristics of an axial fan depend largely on the aerodynamic design and number of the impeller blades together with the angle they present to the approaching airstream stated as angle of attack.

The operation of any turbomachine is directly dependent upon changes in the working fluid's angular momentum as it crosses individual blade rows and this is called as rotor momentum. From the consideration of flow analysis, forces exerted within the individual rows can be visualized as a cascade of plates. Three-dimensional flow is a complex one and can be more manageable with in blade rows as a superposition of number of two dimensional flows, which gives a good blade design and a niche of profile selection. When it comes to axial rotors with hub and casing, it is practical to assume as the stream surfaces at the entry to the annulus which is cylindrical as they move forward through the machine. Each cylindrical meridional stream surface will then intersect the blade row to form a circumferential array of blade shapes known as cascade [7]. This is illustrate in figure 2 adapted from Lewis.

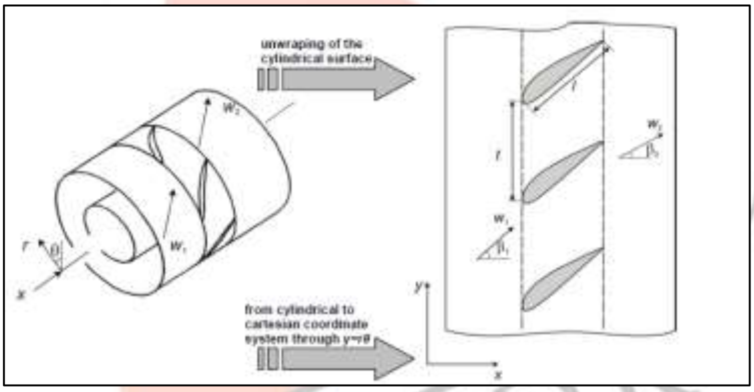


Fig. 2 Development of a cylindrical blade-to-blade section into an infinite rectilinear cascade [7].

This as a three-dimensional flow could then be modelled by a series of such plane two-dimensional cascades, one for each of the cylindrical meridional surfaces equally spaced between hub and casing. We require a cascade of infinite extent for a complete 2-D flow. These cascade should be limited with size with care at mean line interface which is critical and our flow measurements will be made through it, that gives approximately a two dimensional flow.

B. GOVERNING EQUATIONS

The 2D visualization of the flow in a fan can be studied and better understandable by using 2D velocity triangles which will depict the velocity vectors in a fan and the relative velocity angles. From the Figure below many terms can be defined. Two forces can be recognized as acting upon the profile: one in the axial direction (F_{ax}) and the other in the tangential direction (F_t).

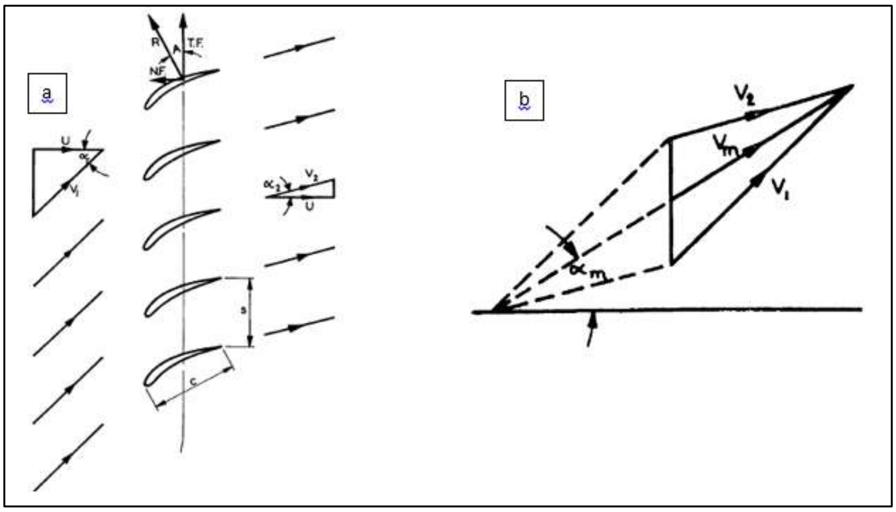


Fig. 3: a) Flow through two dimensional cascade b) velocity vectors (adapted from (3))

The resultant angle A is much identical to α_m, and is given by,

$$\tan \alpha_m = 0.5\,(\tan \alpha_1 + \tan \alpha_2)$$

The next assumption follows that the total head remains constant which is an ideal, non-viscous fluid and hence there is zero drag acting on the aerofoils. The lift force must then be equal to the resultant force and therefore act at right angles to the mean vectors. So, the lift can then be written as follows;

$$L = 0.5 * \rho\,.\,c.\,C_L.\,V_m^2$$

Considering that the component of lift is parallel to cascade, coefficient of lift can be written as follows;

$$C_l = 2\left(\frac{s}{c}\right)\cos \alpha_m\,.\,(\tan \alpha_1 - \tan \alpha_2)$$

C. FAN THEORY

Theoretical head rise w.r.t present design considerations could be written as;

$$\frac{H3 - H0}{0.5\,\rho.\,{V_a}^2} = k_h - k_R$$

Swirl coefficient which is a measure of rotor torque can be defined as the ratio of the swirl velocity and the axial velocity at a given radius.

$$\varepsilon = \frac{V_\theta}{V_a}$$

Flow coefficient can be defined as, the ratio of the axial velocity and rotational rotor speed at a given radius.

$$\lambda = \frac{V_a}{\Omega.\,r}$$

So, the theoretical pressure rise coefficient can be given as;

$$K_h = \left(\frac{2}{\lambda}\right).\,\varepsilon$$

The relative velocity and absolute velocity factors play key role in estimating pressure ratios, blade angle while designing a blade. As per the free vortex theory, the axial velocity component is constant throughout the fan annulus. There is no radial velocity component and the pressure rise is constant in radial direction [Wallis].

Axial velocity is calculated from the continuity equation.

$$V_a(u) = \frac{A_{fan}\,.\,V_{fan}}{A_{annulus}} = \frac{Q_{fan}}{\frac{\pi}{4}.\,(D_{fan}^2 - D_{hub}^2)}$$

The flow over the rotor blades can be represented by either absolute velocities or relative velocities. The schematics of velocity vectors for relative and absolute velocities are given in figure 4.

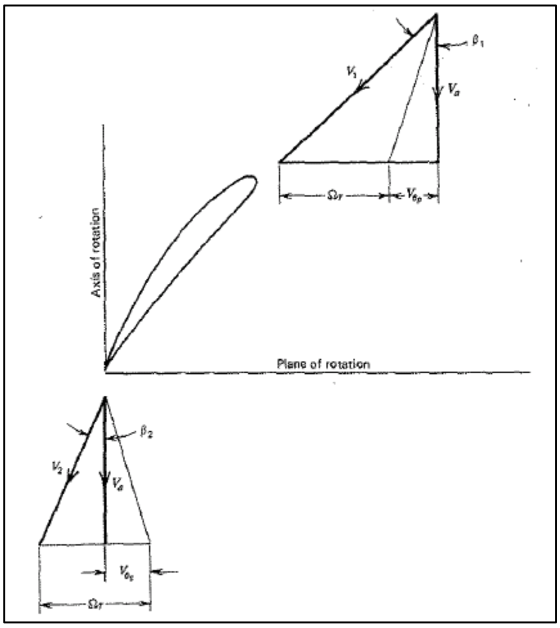


Fig. 4 Absolute velocity vector [6]

The relative flow angles with respect to the rotor blades are calculated by using equations;

$$\beta 1 = \operatorname{atan}\left(\frac{1}{\lambda}\right)$$

$$\beta 2 = \operatorname{atan}\left(\frac{1-\varepsilon}{\lambda}\right)$$

$$\tan(\beta m) = 0.5(\tan(\beta 1) + \tan(\beta 2))$$

The thrust and torque exerted on the flow by the rotor blades can be calculated by using the following equations. The torque acting on the rotor shaft can be expressed in terms of the swirl momentum added to the stream.

$$T = T_c * 0.5\ (\rho\ .u\ .\pi\ .r_{tip}^3)$$

$$T_c = 4\ .\frac{x^3}{3}\ .\varepsilon$$

The main interest in design with the thrust produced by the rotor is in relation to the design of thrust bearings and supports. This could be compensated with an estimate based on the pressure rise across the rotor and the swept area.

$$T_h = T_{hc} * 0.5 * (\rho\ .u^2\ .\pi.r_{tip}^2)$$

$$T_h = \frac{\Delta p.x.r}{0.5.(\rho.u^2)}$$

III. Design details

The numerical investigation of the aerodynamic characteristics of systems that employ axial flow fans for cooling and other duties is an attractive alternative when considering the often prohibitive nature of experimental investigations due to economic and time constraints.

Specifications of the kiln shell cooling fan drawn from an industrial case study as, requirement of volumetric flow to be delivered by each fan is 5 m3/s with a system static pressure of 30 mm WG. Density was set to 1.07 Kg/m3 as per site elevation and temperature, motor efficiency assumed in preliminary design as 89% based on electrical motor chosen as per Siemens catalogue.

Speed was set to be 1450 rpm against 1500 rpm of previous design based on specific speed factor discussed in the chapter-2. Using Bleier's background charts [4] diameter of a fan was finalized to be 800 mm. Number of blades are chosen as 6.

As per the specifications drawn from the given conditions, speed and type of the fan can be. This preliminary design can be illustrated as an estimate for a system curve with a performance curve, which was shown in figure 5.

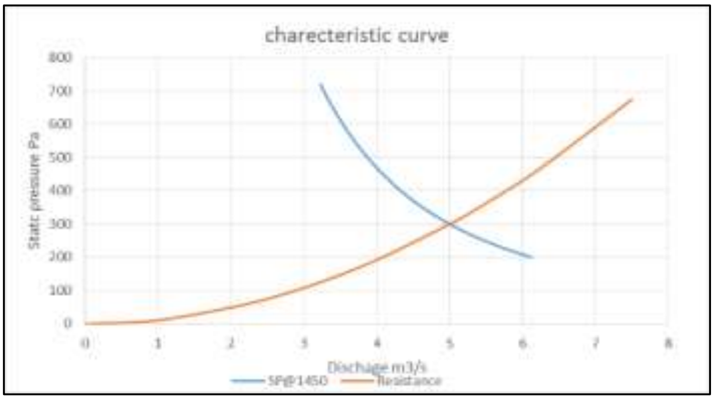
Table -1 Specifications

Specification of a fan	
Volumetric flow, m^3/s	5
Static pressure, Pa	300
Density, Kg/m^3	1.07
Speed, rpm	1450
Fan diameter, m	0.8

For obtaining best combinations of the overall unit preliminary designs were carried out with 5 possible combinations of hub to tip ratio. This data of preliminary calculations was used in reduction of the given data to a non-dimensional form suitable for detailed design of a blade. It can be ascertained for the study of loading on the rotor blades especially at the blade root which is the most critical station. This preliminary design is also utilized for estimating component losses which determines the overall efficiency.

Fig. 5 System characteristic curve of Preliminary design with assumptions

Preliminary design data was provided in appendix. Analysing the design data, hub to tip ratio was chosen as 0.5 which gives, an



aspect ratio approximately to 2, hub diameter as 400mm with static efficiency of 71%. This process is considered as mean line theory in the literature.

The efficiency loss due to profile drag Cdp, were aimed at keeping either the blade element efficiency or the lift coefficient constant along the blade.

With the above figure 6, it is quite evident that the camber plate is less efficient in work to the airfoil (C4). Drag is so high with lower lift coefficients in the camber plate which results in decrease of maximum lift-drag ratio. So airfoil profile is providing an advantage to improve the efficiency over camber plate.

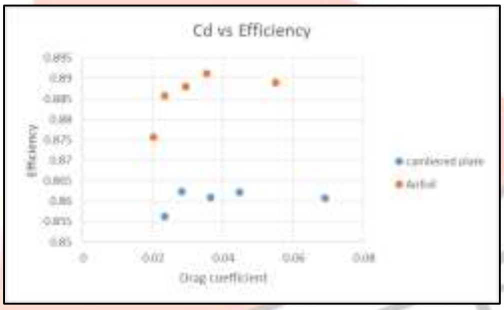


Fig. 6 coefficient of drag vs efficiency

Airfoil selection is primarily based on Reynolds number, thickness ratio, and chord length, maximum camber in chord percentage, lift and pressure coefficients. 30 foils were considered from which ten have been selected for design of blade, out of which 2 are chosen for test. Finally two foils are selected, C4 and NACA 58106. Later based on CFD studies, a 300 camber C4 foil with leading edge radius interpolated with NACA 5 series nose radius gave a fine aerodynamic result. The sample coordinates of it are presented in appendix. Java foil and design foil software are used while selecting aerofoils.

A sample airfoil model has been presented in figure 7.

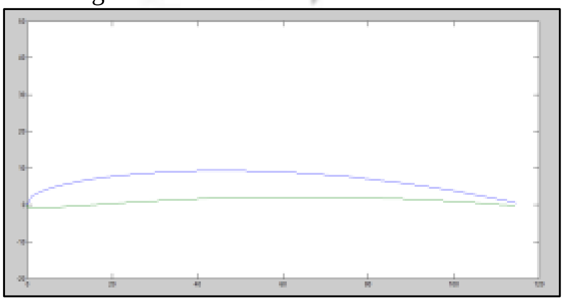
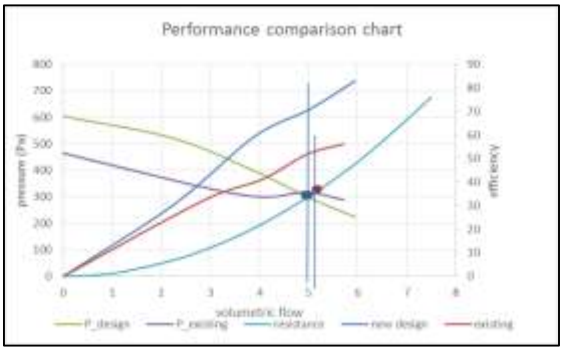


Fig. 7 Airfoil profile shape

Performance characteristics of the design had been compared with existing fan and presented in figure 8.

Fig. 8 Performance curves for comparison purpose



In figure 8, a dot in blue colour represents the operating point of new design with increase in efficiency and performance, and the dot in brown colour represents existing operating conditions which lacks in efficiency comparatively.

IV. CFD Simulation

In order to test the flow around the blade profile, isolated profiles were simulated to carry out the flow analysis over the investigated airfoil, so that the aerodynamic performance of the single profile was snapped and presented. The thickness of the profile was chosen to be around 8 mm and the flow was investigated on the center line of profile, where the influence of side walls were kept with no significant effect on the simulated model.

Boundary conditions are key to play a simulation for an appropriate result that should be satisfactory. So, care has been taken while considering boundary conditions. At the inlet section, the specified flow velocity which is nothing but an axial velocity acting over a meridional section of a blade profile, which is calculated to be 16 m/s.

Outlet of the domain was open boundary, which is true with the case to have an ambient pressure. On to the lateral surfaces, symmetry boundary has been set, over to the top and bottom of the domain is also a symmetry boundary. As up on observation, after a height of the domain of 20 times the chord length from top to bottom, the difference in results between applying symmetry and / or opening boundary will sought to be negligible. The simulations were carried out for viscous flow with air as ideal gas.

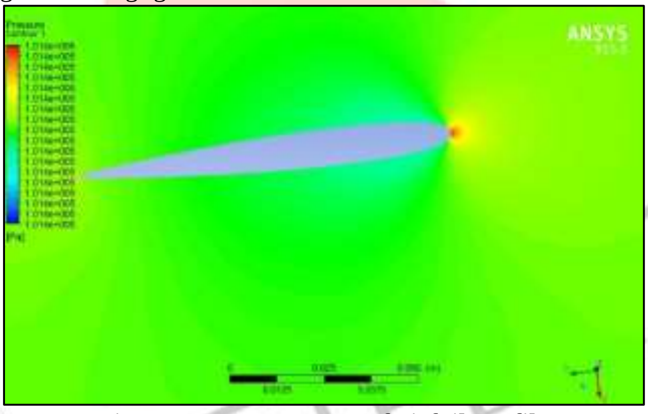


Fig. 9 pressure contour of airfoil profile

To bring the simulation results closer to real-life operating conditions, to a CAD model at the inlet of the impeller a rig section which is long enough that the flow entering the impeller domain can be considered as a fully developed with the 3 Dpipe, and at the outlet where an ambient condition prevails another pipe with a length of 2 Dpipe was included. Its pictorial depiction was shown in below figure 10.

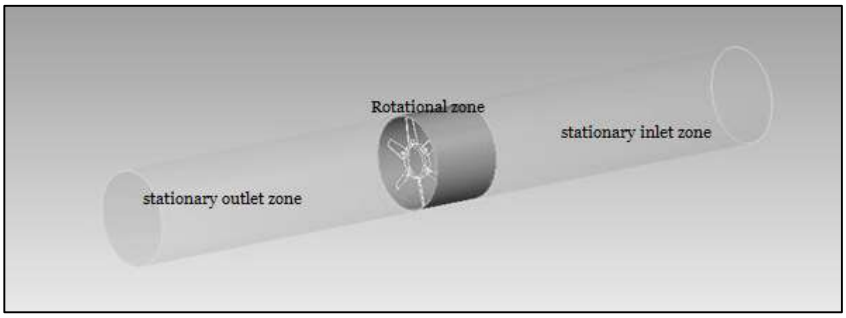


Fig. 10 CAD model that represents domains

The inlet boundary with specified velocity was applied to the rig domain, flow direction normal to the boundary condition and medium turbulence intensity (5%). A rotational speed of 1450 rpm was applied to the fan domain. The outlet boundary was applied to the ambient domain by specifying a 250C temperature with an air density of 1.07 kg /m^3.

There are two flow zones described for the full model. One zone is the fluid where the flow is axial and the other zone is fluid where the flow is rotational with respect to the rotor. The rotational fluid zone is defined by a moving reference frame. Zones are shown in Figure 10.

Comparative efficiency curves of both analytical and CFD had been presented in figure 11.

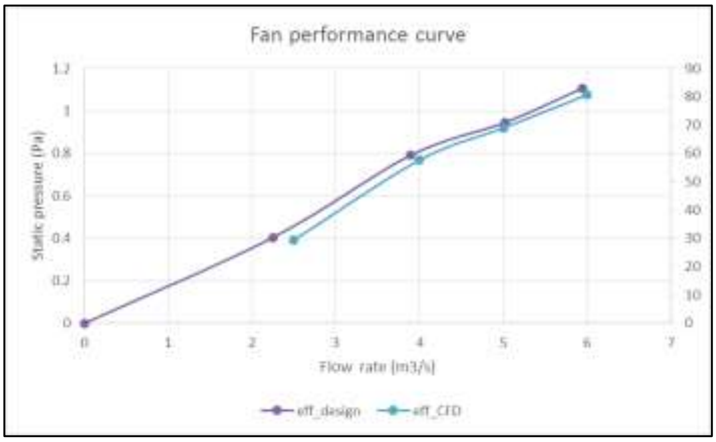


Fig. 11 efficiency of both CFD and analytical models

Figure 11 demonstrates that the design is evaluated with CFD validation which are quite close and in acceptable margins. Hence the designed model is validated for physical construction and experimental evaluation.

V. CONCLUSIONS

Efficiency has been increased when compared with the existing machine from 52% to 63% of total to total efficiency. Performance of the new design was verified with commercial CFD software, ANSYS CFX, which is in good agreement.

Analysis of aerofoils gave a good experience for processing and analysis while constructive design of blades of turbomachinery.

It can be concluded from the analysis that the design meet the requirement and true with the case that could be implemented for practical and functional case.

VI. References

[1] "Fans", Bruno Eck, Pergamon Press.
[2] "Fans & Ventilation - A Practical Guide"; W T W (Bill) Cory, Elsevier publishers, 2005.
[3] "Axial flow fans, design and practice", R.A.Wallis, 1961.
[4] "Fan Handbook", Franck P Bleier, 1998.
[5] "The fan including the theory and practice of centrifugal and axial fans", Charles H. Innes.
[6] "An Engineering Approach to Blade Designs for Low to Medium Pressure Rise Rotor-Only Axial Fans", R.A.Wallis et.al, CSIRO Experimental Thermal and Fluid Science, 1993.
[7] "Turbomachinery performance analysis", R.I Lewis, 1996.
[8] "A theoretical derivation of the Cordier diagram for turbomachines"; Epple et.al, Journal of Mechanical Engineering Science, 2011.
[9] "Axial fan design: modern layout and design strategy for fan performance optimization" by Maria Pascu
[10] "The Design and Performance of an Axial-Flow Fan"; Lionel s. Marks et.al, AER-53-13, transactions of the american society of mechanical engineers, 1934.
[11] "A Comparison of Two Methods for Predicting the Potential Flow around Arbitrary Airfoils in Cascade"; D. Pollurd and J. Wordsworth, 1963.
[12] "Fluid machinery: application, selection, and design"; Gerhart and Wright, 2nd edition, 2009.
[13] "Airfoil Aerodynamics Using Panel Methods" by Richard L. Fearn
[14] "Centrifugal and axial fan applications and design criteria" by I.Howitt
[15] "Applied computational fluid dynamics techniques"; Rainald Löhner, 6th edition, 2008.
[16] ANSYS CFX Reference Guide.