© AUG 2019 | IRE Journals | Volume 3 Issue 2 | ISSN: 2456-8880

Numerical Study on Flow Characteristics of Radial Tipped Impeller for Centrifugal Fan

ZAR CHI THAUNG[1], SWE ZIN NYUNT[2], KYI KYI SWE[3]

[1] *Mechanical Engineering Department, Technological University (Maubin), Myanmar*
[2] *Mechanical Engineering Department, Technological University (Thanlyin), Myanmar*
[3] *Mechanical Engineering Department, Technological University (Thanlyin), Myanmar*

Abstract- In this paper, single-stage centrifugal fan is observed by numerical study on the flow characteristic. The objective of this paper is to study the flow characteristics in the centrifugal fan under variable inlet parameters by numerical analysis. Numerical study also focuses on the flow velocity, temperature and pressure distributions around the fan due to the effect of inlet radial tipped impeller by applying CFD software ANSYS CFX-15.0. The model of radial tipped centrifugal fan is created by Solid Works software. The flow simulation has been carried out the Reynolds Averaged Navier - Stokes equation for incompressible flow is solved with standard eddy-viscosity k-epsilon turbulence model to study the properties of the fan.

Indexed Terms - Centrifugal fan, Numerical study, Inlet parameters, CFD software, Flow characteristic

I. INTRODUCTION

Centrifugal fan is one of the most important turbo machines which widely used in present industrial and domestic life [1]. The centrifugal fan uses the centrifugal power generated from the rotation of impellers to increase the pressure of air when the impeller rotates. The gas near the impeller is thrown-off from the impellers due to the centrifugal force and then moves into the fan casing. As a result the gas pressure in the fan casing is increased. The gas is then guided to the exit via outlet duct as shown in Fig.1. Centrifugal Fan consists of mainly two parts, namely, the volute casing and the impeller [2].

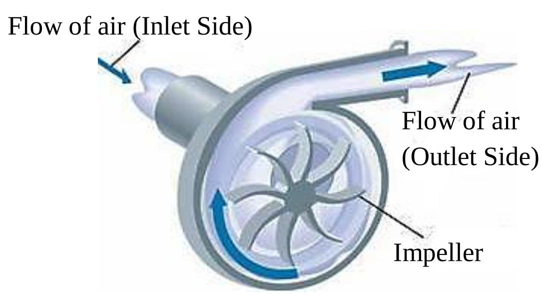

Figure 1. Fluid flow in an impeller [3]

An impeller is a rotor inside a tube or conduit used to increase the pressure and flow of a fluid [4].The pressure rise and flow rate is depend on the peripheral speed of impeller and blade angles. There are three common types of centrifugal fan. Each type is shown in diagrammatic form below [5].

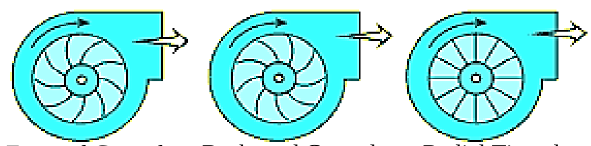

Figure 2. Three common types of fan [6]

II. DESIGN METHODOLOGY

This work is based on an industrial requirement for Fume Extraction Fan of texturizing machine. The input parameters for the radial tipped centrifugal fan from texturing machine are summarized below.

Table I Input parameters of centrifugal fan

Descriptions	Symbols	Values	Units
Rotational speed	N	1450	rpm
Air flow rate	Q	0.91	m^3/s
Inlet air temperature	T_a	30	˚C
Inlet air pressure	P_a	101.33	kPa
Pressure ratio	ε_p	1.0059	-
Shaft diameter	D_s	12	mm

A. Design of Impeller

This design procedure is based on the fundamental principles of fluid flow with continuity and energy equations. The design follows the path from suction to discharge. The design parameters are calculated with equation (1) through (20). Design procedure and calculations are presented below:

1) Calculation of the input power (P)

The input power of the fan gives the power required at the fan shaft with varying quantity of air delivered [7]. To calculate the input power, air flow rate and total adiabatic head must be known. The input power is determined from the relationship

$$P = \frac{\rho_a \, g \, Q \, H_{ad}}{\eta_o} \tag{1}$$

The value of overall efficiency is assumed as 75% [6].

Total adiabatic head is

$$H_{ad} = \frac{RT_a}{\frac{k-1}{k} \times g}\left(\varepsilon_p^{\frac{k-1}{k}} - 1\right) \tag{2}$$

2) Calculating the Hub and Impeller Eye Diameters

The diameter of hub is given as;

$$D_h = D_s + (19.05 \text{ to } 50.88) \tag{3}$$

Impeller eye diameter is calculated by;

$$Q_{real} = 1.1\ Q \tag{4}$$

$$D_0 = \sqrt{\frac{\pi}{4} \times \frac{Q_{real}}{V_0} + D_h^2} \tag{5}$$

The air velocity at impeller eye is the range between 0.2 to 20 m/s. So, the value of V_0 is taken as 20 m/s [9].

3) Calculation of Impeller Inlet and Outlet Diameters

The impeller inlet diameter may be slightly two percent greater than the impeller eye diameter [10].

$$D_1 = 1.1\ D_0 \tag{6}$$

The outlet diameter of impeller is given as;

$$D_2 = \frac{60}{\pi N} \times \sqrt{\frac{g H_{ad}}{K'}} \tag{7}$$

The coefficient of the friction and turbulence in the impeller is the range 0.5 to 0.65. Taking coefficient, K′ = 0.65 [11].

4) Calculating the Inlet Velocity (V_1)

Impeller inlet tip speed is $U_1 = \frac{\pi D_1 N}{60}$ (8)

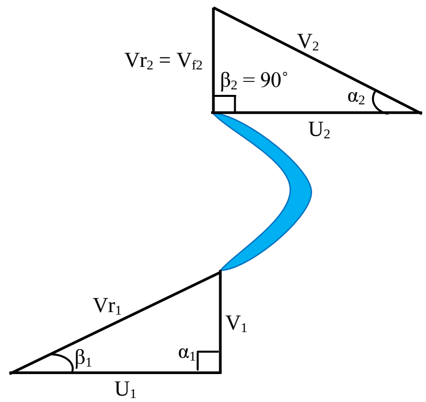

Figure 3. Inlet and Outlet velocity triangle of radial fan

For inlet relative velocity, V_{r1} is slightly greater than V_0 [13].

By using above diagram, the absolute inlet velocity is

$$V_1 = \sqrt{V_{r1}^2 - U_1^2} \tag{9}$$

5) Describing the Number of Blades (Z)

An impeller is a rotating component equipped with blades used in turbo machinery.

The number of blades used varies 6 to 16 for radial blade [12].

To find the blade inlet angle β_1 by using above figure.

$$\tan(\beta_1) = \frac{V_1}{U_1} \tag{10}$$

For radial blade, β_2 is 90˚. So, the number of blades is $Z = 6.5 \times \frac{D_2 + D_1}{D_2 - D_1} \sin\left(\frac{\beta_1 + \beta_2}{2}\right)$ (11)

6) Calculating the Outlet Velocity (V_2)

To find the outlet velocity, impeller outlet tip speed and radial outlet velocity must be known.

Impeller outlet tip speed is $U_2 = \frac{\pi D_2 N}{60}$ (12)

The radial outlet velocity, V_{r2} should be 15 % slightly less than radial inlet velocity, V_{r1} [14].

So, absolute outlet velocity, $V_2 = \sqrt{V_{r2}^2 + U_2^2}$ (13)

7) Describing the Impeller Inlet and Outlet Widths

Impeller inlet width is $b_1 = \frac{A_1}{\pi D_1 \varepsilon_1}$ (14)

The inlet vane thickness factor (ε_1) that value is chosen as 0.925.

Outlet vane thickness factor, $\varepsilon_2 = \frac{\pi D_2 - \frac{Z \times t}{\sin(\beta_2)}}{\pi D_2}$

Assume the impeller blade thickness, t = 4.32 mm.

By using continuity equation,

$$Q_{real} = A_2 \times V_{f2} \quad (16)$$

Impeller inlet width is $b_2 = \frac{A_2}{\pi D_2 \varepsilon_2}$ (17)

8) Calculating the Impeller Outlet Pressure (P_2)

$$P_2 = P_0 \times \varepsilon_p \quad (18)$$

Pressure at impeller eye, $P_0 = \frac{P_a}{\varepsilon_{p(eye)}}$ (19)

9) Calculating the Impeller Outlet and Eye Temperature

Impeller Outlet Temperature, $T_2 = T_0 \times \varepsilon_p^{0.286}$

Temperature at impeller eye, $T_0 = \frac{T_a}{\varepsilon_{p(eye)}^{0.286}}$ (21)

10) Calculating the mass flow rate of the air (m°)

$$m^\circ = \rho_a A_2 V_{f2} \quad (22)$$

According to above equations, calculated results for radial tipped impeller of centrifugal fan are clearly expressed in the following table.

Table II Results data of radial tipped impeller

Descriptions	Symbols	Values	Units
Hub diameter	D_h	31	mm
Eye diameter	D_0	199	mm
Inlet diameter	D_1	218.9	mm
Outlet diameter	D_2	369	mm
Inlet width	b_1	132	mm
Outlet width	b_2	48	mm
Number of blades	Z	14	-

Table III Inflow and outflow results of impeller

Descriptions	Symbols	Values	Units
Peripheral Velocity at Inlet	U_1	15.03	m/sec
Relative Velocity at Inlet	V_{r1}	19.5	m/sec
Absolute Velocity at Inlet	V_1	10.8	m/sec
blade inlet angle	β_1	44	degree
Flow Velocity at Outlet	V_{f2}	16.58	m/sec
Peripheral Velocity at Outlet	U_2	28.69	m/sec
Relative Velocity at Outlet	V_{r2}	16.583	m/sec
Absolute Velocity at Outlet	V_2	33.14	m/sec
blade inlet angle	β_2	90	degree
Mass flow rate	m°	1.17	Kg/sec
Outlet Pressure	P_2	101.83	kPa
Outlet Temperature	T_2	305.1	K

III. NUMERICAL STUDY

In order to introduce the subject of fan analysis, this research emphasizes the numerical approach towards the prediction of the flow field in fan. To understand the importance of employing advanced numerical methods for analyzing fan flows a thorough discussion of the general characteristics of the flow field is provided three dimensional analyses [15].

A. Characteristics of the Three Dimensional Flow Field

The flow in a radial tipped fan is three - dimensional and should be treated as such in order to achieve a solid knowledge of the flow field and understand the mechanisms causing the performance do diverge from the ideal case. An insight into the physics of the flow can be derived through a discussion with special attention to secondary and partial load phenomena. When looking into the flow pattern in radial tipped fan prompt it is essential to understand the influence of the forces acting upon the flow [16].

B. Basic Governing Equation of the Model

Computational Fluid Dynamics (CFD) is concerned with numerical solution of differential equations governing transport of mass, momentum and energy in moving fluids [17]. The equations used in this numerical flow analysis are the Reynolds Averaged Navier Stokes (RANS) equation with turbulence models based on eddy viscosity concept.

1) Mass conservation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_j}(\rho U_j) = 0$$

2) Momentum conservation

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_i U_j\right) = -\frac{\partial p'}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] + S_M$$

Where the sum of body forces S_M, is the effective viscosity accounting for turbulence and is the modified pressure [18].

IV. CFD METHODOLOGY

In order to obtain better design in Computational Fluid Dynamics (CFD), following procedure is applied in Fig.4, so that the fluid flow can easily be modeled. Initial design of the model is a planning decision and the geometry is generated depending on these initial design considerations, using either CFD modeling tools or other design tools. The first task to accomplish in a numerical flow simulation is the definition of the geometry, followed by the grid generation. This step is the most important step for the study of an isolated impeller assuming an axis symmetric flow simplifies the domain to a single blade passage [17].

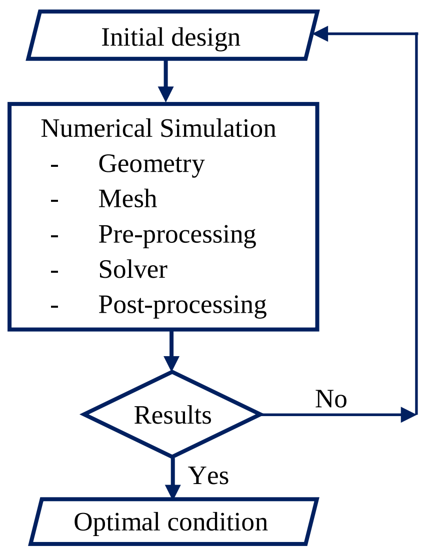

Figure 4. Flow chart showing CFD solving methodology

A. Geometric Modeling

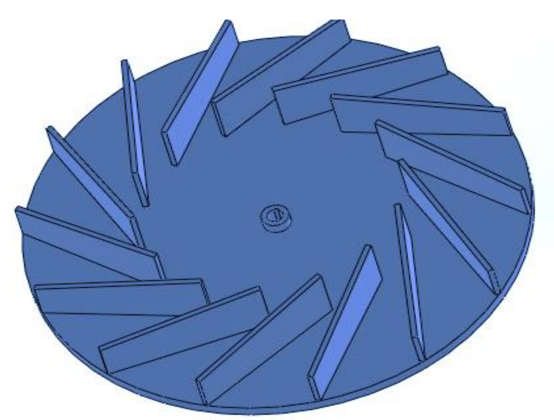

Figure 5. Geometric model of the radial tipped fan

The geometry of design needs to be created from the initial design. Before the modeling of radial tipped impeller, a generalized parameter is calculated for the radial tipped impeller of fan [19]. The design is based on the design methodology discussed in the previous article. The radial tipped impeller of fan is designed in ANSYS Design Modeler by Solid Works 2014.

B. Mesh Generation

The geometry is divided into small sub parts for CFD analysis called mesh and the process is called meshing. The shape of the mesh elements can be triangular, quadrilateral, tetrahedral, hexahedral or prism depending upon the size and shape of the geometry [20]. The shape and size of the mesh elements can be varied and are kept according to the dimension of the geometry, required accuracy and computational power of the system. Representations of the different surface meshes that take part in the study are depicted in the following figure.

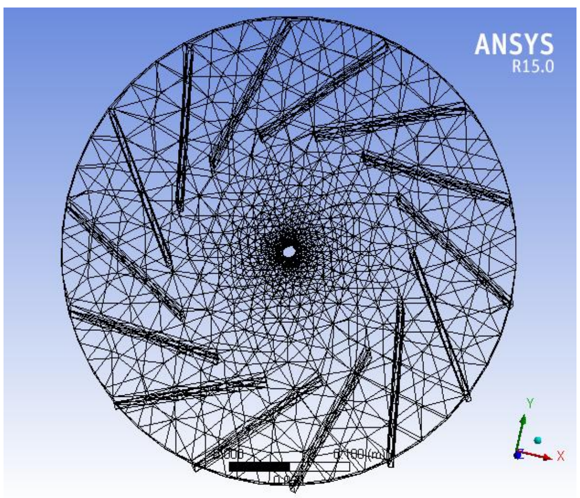

Figure 6. Meshing of flow domain

The summary of meshing data for each surface has been described in Table III.

Table III Mesh number of elements

Domain	Nodes	Elements
1	2753	10647

C. Boundary Conditions

Boundary conditions determine largely the characteristics of the solution to obtain. The first step in Pre-processing is setting up the boundary conditions. Therefore, it is important to set boundary conditions that accurately reflect the real situation to obtain accurate results.

Centrifugal fan geometry consists of three different zones: the inlet region, the impeller and the blades exit. The inlet region and the blades exit are considered as a stationary frame, whereas the impeller zone is studied as a rotating frame. Surfaces between the entry zone and surfaces corresponding to the blades entry and surfaces corresponding to the blades exit are defined as interfaces.

The boundary conditions and interface properties are prescribed in present simulation as shown below.

Table IV Boundary physics for CFX

Domain	Boundaries	
Domain 1	Boundary - inlet	
	Location	inlet
	Flow Regime	Subsonic
	Mass Flow Rate	1.0680e+00 [kg s^-1]
	Turbulence	Medium intensity and eddy viscosity Ratio
	Boundary – outlet	
	Location	outlet
	Flow Regime	Subsonic
	Relative Pressure	1 [atm]
	Boundary - wall	
	Location	wall
	Mass And Momentum	No Slip Wall
	Wall Roughness	Smooth Wall

Figure 7 shows the inlet and outlet boundary conditions and interface conditions.

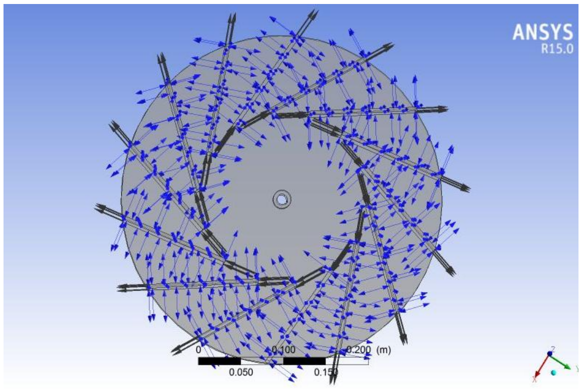

Figure 7. Boundary conditions of impeller for CFX model

D. Solver Control in CFX Model

The iterative process is repeated until the change in the variable from one to the next iteration becomes so small that the solution can be considered converged. The convergence of the simulations is said to be achieved when all the residuals reach the required convergence criteria. The convergence criterion for the root mean square (RMS) residual is set to 10^{-6} for the termination of iterations. The solver was run for 200 iterations. For steady-state problems, the CFX-Solver uses a robust, fully implicit formulation so that relatively large time steps can be selected. A steady-state calculation will usually require between fifty and hundred Physical Time steps to achieve convergence as shown in fig.8.

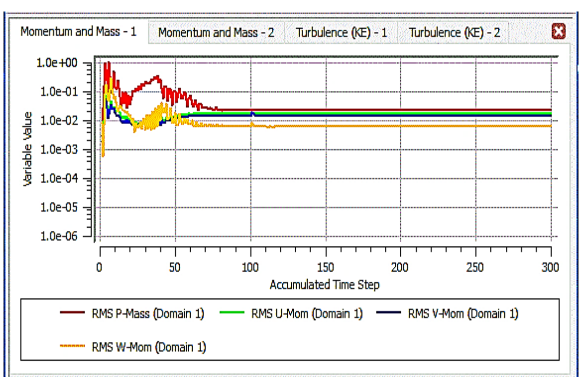

Figure 8. Graph of mass and momentum during simulation against number of iteration

V. SIMULATION RESULTS FOR RADIAL TIPPED IMPELLER

The flow simulation has been carried out for constant cross-sectional area of the impeller and mass flow rate. According to the simulation result, the velocity, temperature and pressure distributions are shown in Figure 9, 10 and 11.

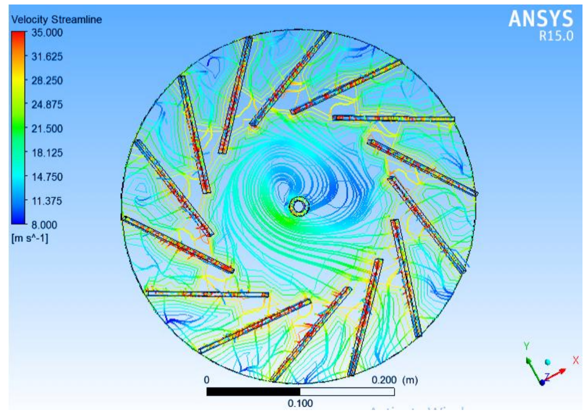

Figure 9. Velocity distribution on the radial tipped impeller

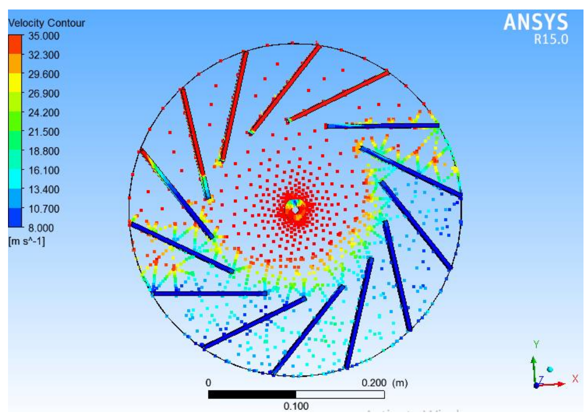

Figure 10. Velocity contour on the radial tipped impeller

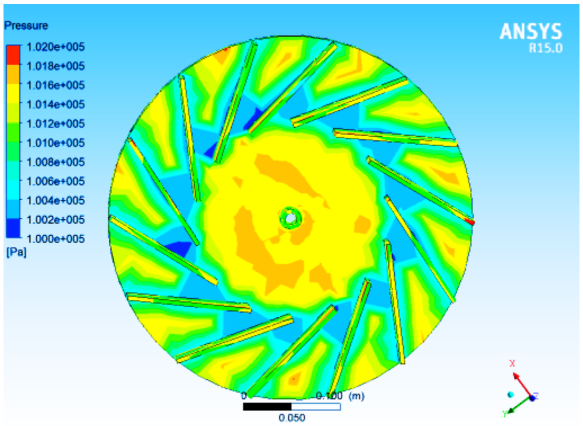

Figure 11. Pressure distribution on the radial tipped impeller

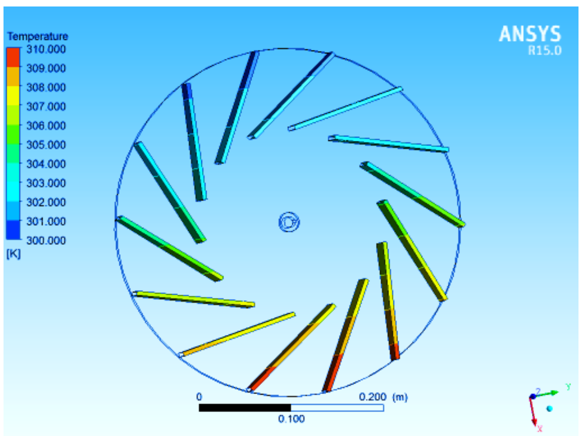

Figure 12. Temperature distribution on the radial tipped impeller

According to Figure 9 and 10, the impeller inlet and outlet velocity distributions are from 8 to 35 m/s. From the side of pressure distribution diagram, pressure distribution is within the range from 100 to 102 kPa. With respect to the pressure distribution analysis, impeller eye pressure is less than the atmospheric pressure. The inlet temperature distribution at the lowest, the outlet temperature near 310 K is very close to the outlet theoretical value. Finally, the variations of the pressure contour, temperature, velocity streamlines and velocity contour respectively on the surface of the impeller could be observed. By observing the numerical simulation result, the three variations for impeller are close to theoretical value.

VI. CONCLUSIONS

In this research, single-suction centrifugal fan with a radial tipped impeller is used. A numerical model of an impeller has been successfully generated and the complex internal flow fields are investigated by using ANSYS - CFX 15.0. Finally, theoretical results are verified by comparing with simulation results. This result showed that it was technically possible to use in the industrial requirement for Fume Extraction Fan of texturizing machine.

ACKNOWLEDGMENT

The author takes opportunity to describe her gratitude to those who have helped her directly or indirectly in writing this paper. The author is grateful to her beloved parents, sister and brothers who have always given great help, encouragement, sacrifice and support throughout her life.

REFERENCES

[1] Golla J.Yothi, A.Vinutha, The Effect of Impeller Blade Design on Centrifugal Blower- CFD Approach, (IJITR), Volume No.5, Issue No.4, June – July 2017.

[2] Vivek Brahmbhatt, Gaurav Patel, Performance Enhancement of IND 25 Centrifugal Fan by CFD, International Journal of Innovative Research in Science, Engineering and Technology, Vol. 3, Issue 3, March 2014.

[3] Turbines, compressors and Fans by S. M. Yahya.

[4] Fluid Mechanics and thermodynamics of turbo machinery by S.L. Dixon.

[5] David J. Gingery, Centrifugal Fans for the Home Shop, Copyright 1987 by Lindsay Publications, Inc.

[6] Austion H.Church, Centrifugal Pump and Blowers, John Wilely and Sons, Inc,New York, 1972.

[7] Banerjee S.P., Mine Ventilation, Dhanbad, India, 2003.

[8] R.S. Khurmy, J.K. Gupta ,A Text Book of Machine Design, Fourteenth Edition, Ram Nagar, New Delhi, 2005

[9] Frank P. Bleier, P.E, Fan Hand Book, Selection, Application and Design, McGraw-Hill Companies, 1998.

[10] Krivchenko, G.I.: Hydraulic Machines: Turbines and Pumps. Mir Publishers, Moscow, (1986).

[11] Balje, O.: Turbo machines: A Guide to design, selection and theory. Published by John Wiley and Sons, New York, United States of America, ISBN 10: 0471060364, 1981.

[12] Shepherd D.G.: Principles of Turbo machinery, MacMillian Publishing Co., Inc, New York, 1956.

[13] Vineet Ahuja, Ashvin Hosangadi, Impeller Design of a Centrifugal Fan with Blade Optimization, International Journal of Rotating Machinery, July 2011.

[14] Manjunatha, Nataraj J R, Design and Analysis of Impeller Blade for Axial Flow Pumps, international journal of engineering researches and management studies, may 2015.

[15] Younsi M., Bakir F., Kouidri S.,Influence of impeller geometry on the unsteady flow in a centrifugal fan: numerical and experimental analyses, International Journal of Rotating Machinery, 2007.

[16] Thomas Bouquet, Farid Bakir, Study of the 3D flows in the forward-curved blades centrifugal fans, ASME Joint U.S. - European Fluids Engineering Division Conference (FEDSM), Canada, 2002.

[17] Anil W. Date, Introduction to Computational Fluid Dynamics, Cambridge University Press, 2005.

[18] M. Younsi, F. Bakir, S. Kouidri, Influence of design parameters on the unsteady flow in a centrifugal fan, ASME/JSME 5th Joint Fluids Engineering Conference (FEDSM), San Diego, California, USA, 2007.

[19] Jeon, Wan-Ho, A numerical study on the effects of design parameters on the performance and noise of a centrifugal fan, Journal of Sound and Vibration, Vol 265, pp. 221-230, 2003.

[20] Anonymous: ANSYS CFX Solver Modeling Guide, 2013.