An Experimental and Numerical Investigation of Volute Tongue Clearance Variation on Performance of Centrifugal Blower

Nitin S. Jadhav, PG Student -Mechanical Design Engineering, AISSMS COE, SPPU, Pune.
S.R.Patil, Assistant Professor -Mechanical Engineering Department, AISSMS COE, SPPU, Pune.

Abstract— **In this paper, the effect of volute tongue clearance on the performance of centrifugal blower with backward-curved blades has been experimentally investigated and compare with numerically simulated data. The casings of the centrifugal blower with different volute tongue clearances of 6 % to 12.5 % of impeller diameter are used for experimentations. As a first step, an experimental setup is developed as per Indian standard IS4894:1987 and the experiments are performed with respective standard. The performance parameters flow rate, total pressure, shaft power and efficiency of the blower is calculated. The results show that the volute tongue clearance has a significant effect on the performance of centrifugal blower. The efficiency and a total pressure of blower increase when the volute tongue clearances are decreased.**

Index Terms - **Centrifugal Blower, CFD, Performance, Volute Tongue Clearance,**

I. INTRODUCTION

THE centrifugal blowers is frequently used in many applications to produce high pressure. It correspondingly develops high-pressure ratios in small axial distance related to axial fans. They are commonly used in heating, ventilation, air conditioning systems and various other applications. At all the individual design, centrifugal blowers could all be defined by reference Fig.1. Depending on the blade design, centrifugal blowers can be classified into forward-curved, backward-curved, radial, and airfoil types [1]. By definition, a blower is a device for moving air which utilizes a power driven the rotating impeller. A blower has at least one inlet opening and at least one outlet opening. The rotating impeller transfers mechanical energy from the shaft to the air stream. The energy in air appears in the form of air velocity and air pressure. It is important to note that all of the energy added to the air is added to power driven rotating impeller. When the air moves into the suction side of the impeller, the shape of the casing provides the necessary pressure to the flowing air. In centrifugal blower, the air was forced to the periphery and discharged into the volute. The outlet is separated mainly of the volute over the volute tongue clearance, which is the point of closest approach among moving and stationary members. Many experimental studies have been investigated on the performance of centrifugal blower a strictly incomplete primary flow, with initial flow separation on the suction side, and poor matching between impeller outlet and volute tongue induced the inefficiency of that blower. The detailed velocity region near the volute tongue of a backward curved centrifugal blower using PIV and drew a conclusion that the matching between impeller outlet flow configuration and volute tongue has a major effect on the flow loss in the blower. Adjust the volute tongue clearance it reduces the reverse flow rate through the volute tongue. The fan performance improved because the reverse air flows at the region near volute tongue are effectively reduced [2]. The variations of the volute tongue clearance in order to decrease the pressure variations in this region. Decreasing impeller-tongue clearance are widely used methods for centrifugal fan performance improvement [3]. Increase the gap between the tongue and the impeller, thus keeping the tongue out of the region with severe nonuniform flows. Although it is known that the performance of a pump depends on this gap [4]. The effect of volute tongue position on the circumferential pressure distribution was observed. The differences in flow pattern and flow reversal are larger when the radial gap is smaller and the blockage effect is reduced with increasing radial gap. Thus, the volute tongues position has a significant effect on the performance.

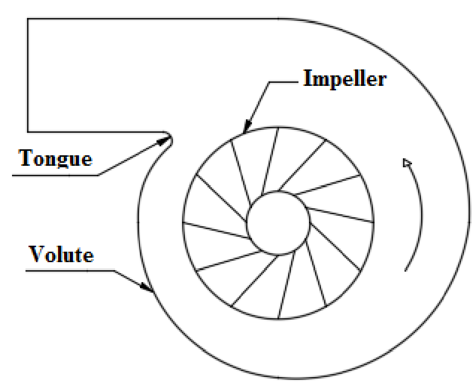

Fig.1.Schematic diagram of the centrifugal blower.

Large pressure variation is observed in the vicinity of volute tongue region and it decreases with increase in distance from the tongue [5]. Computational fluid dynamics (CFD) packages have used to study the turbo-machine and computational fluid dynamics has an effective tool for analysis the centrifugal blower performance. The Navier-Stokes equation by the finite volume method which has been applied widely in fluid mechanics and engineering applications [6, 7]. The modified tongue positions have affected to the power consumption and the total efficiency of the fan. [8]. the volute tongue clearance had a substantial effect on the performance of centrifugal fan [9, 10, 11].

II. EXPERIMENTAL SETUP

A standard test setup for the performance analysis of centrifugal blower according to Indian standard IS4894:1987 is developed. The test setup contains a centrifugal blower and a test inlet duct. The inlet duct has 125 mm in diameter and 552.5 mm in length. The blower is driven by a three-phase electric motor at a constant speed of 2800 rpm. The flow rate can be controlled by orifice plate in the inlet duct during experiments. The test setup is assembled and the experiments conducted in accordance with IS4894:1987. Fig.2 shows a test setup with its main elements and Fig.3 shows the actual test setup. The flow rate through the blower is calculated by the static pressure measured by the U-tube manometer at section 'A' of the inlet duct. The average static pressures at the blower inlet are measured over four taps equally distributed on the circumference of the inlet duct at section 'B' and the outlet total pressure measured at the outlet of the blower for calculating the blower total pressure. The distance from the static taps at blower inlet duct is 125 mm.

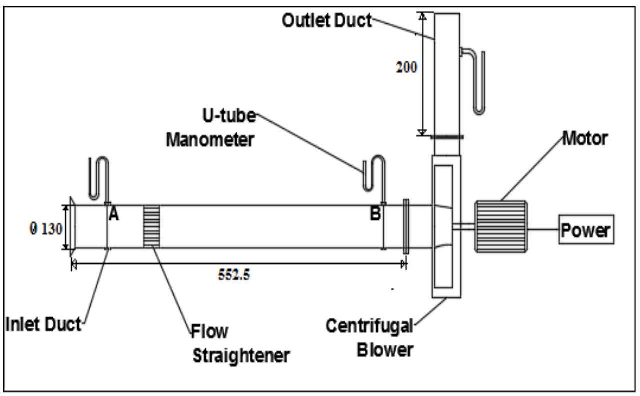

Fig.2. Test setup.

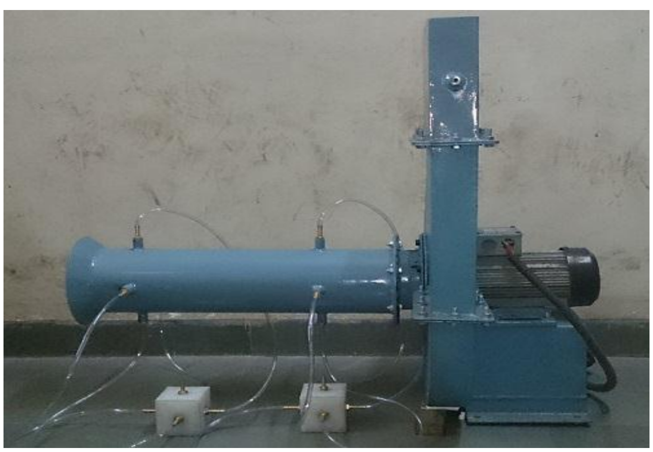

Fig.3. Actual test setup.

The rotation speed of the blower is measured by a hand-held digital tachometer. The input power required to drive the impeller is measured using a digital multimeter. The maximum uncertainties for different instruments are as follows,

Digital tachometer: ± 0.05 % (± 10 rpm)
Digital multimeter: ± 0.01 % (± 0.01 Amp)
U-tube manometer: ± 1 mm of H_2O

III. TESTING OF CENTRIFUGAL BLOWER

An original centrifugal blower and its modified centrifugal blower are used for the experimental measurement. The difference between the original blower and its modified blower is only the volute tongue clearance of blower. Fig.1 shows the sketch of the original centrifugal blower and Table I show all dimensions of the original centrifugal blower. The original centrifugal blower impeller-volute clearance is 12.5% of impeller diameter. The modification is carried out only in the casing of the centrifugal blower, by decreasing the impeller-tongue clearance as shown in Fig.4. The dimensions of the original and three modified volute tongue clearance are listed in Table II. The modified volute tongue clearance of centrifugal blower's are10%, 8% and 6% of impeller diameter and the modified blowers are named as A_1, A_2 and A_3 respectively.

TABLE I
ORIGINAL BLOWER DIMENSIONS

Parameter	Dimensions
Impeller outlet diameter d_o(mm)	280
Impeller inlet diameter d_i (mm)	140
Impeller blade number z	12
Impeller width b (mm)	20
Impeller blade angle (^{0})	30
Casing width w (mm)	65
Volute tongue radius r (mm)	14
Volute tongue clearance (mm)	35
Casing inlet diameter (mm)	125
Casing outlet B×L (mm)	65 × 185

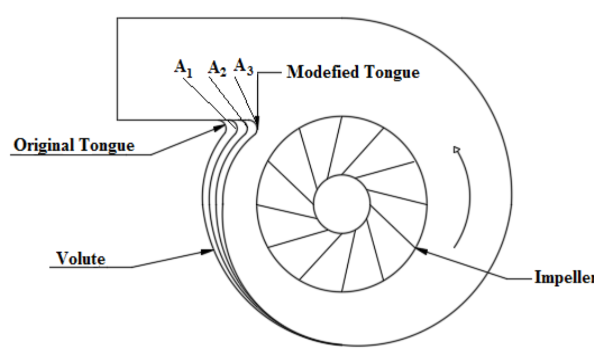

Fig.4. Modified centrifugal blower.

TABLE II
THE DIMENSIONS OF THE ORIGINAL AND THREE MODIFIED VOLUTE TONGUES CLEARANCE

Tongue	t_c/d (%)
Original	12.5
A_1	10
A_2	8
A_3	6

IV. NUMERICAL ANALYSIS

The commercial computational fluid dynamics (CFD) package, fluent is used to simulate backward curved centrifugal blowers. Fluent solves the Navier-Stokes equation by the finite volume method which has been applied widely in fluid mechanics and engineering applications. It has been presented that the fluent quasi-steady simulation can be used to calculate centrifugal blower performance. In ANSYS fluent there are three steps to solve the problem first is to import the model. The three-dimensional centrifugal blower models were first generated in Solid works 2015 software is shown in Fig.5. The next step is mesh generation, the meshing element for rotating impeller were defined by tetrahedral elements, and hex elements were selected for the inlet duct, casing and outlet duct volumes. A typical mesh model is shown in Fig.6.

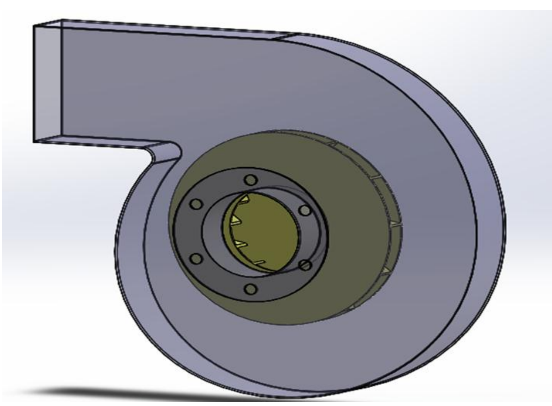

Fig.5. Centrifugal blower model.

In this study, numerous assumptions were made to facilitate the simulations:

1. No-slip boundary condition.
2. Gravity effects are negligible.
3. Fluid properties are not functions of temperature.

A computational model, using the k–ε turbulence model, was used to predict the internal flow of volute. This allows a two-equation model to account aimed at effects of convection and diffusion of turbulent energy. The transported variables are the turbulent kinetic energy, which controls the energy in the turbulence. The subsequently transported variable is the turbulent dissipation which can be thought of as the variable that defines the length scale of the turbulence. The k-ε model has developed one of the industry standard models and is commonly used for most of the engineering problems.

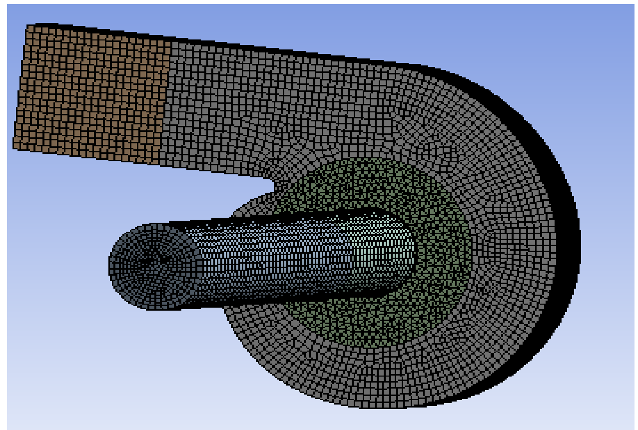

Fig.6. Mesh model.

The last step is boundary conditions for blowers discussed in this study consisted of the inlet, outlet, and impeller wheel boundary conditions. At the beginning of this study, the atmospheric pressure at the entering of the inlet was given as the inlet boundary condition and outlet boundary condition was set as atmospheric pressure. For unsteady state simulations, the multiple reference frames models are used. Surfaces that rotate relatively are defined as “moving wall”. Additionally, as they are dependent on the fluid nearby them and as they rotate, they are defined as “relative to adjacent cell zone” plus “rotational motion”. The turbulent flow is incorporated through LES.

V. RESULT AND DISCUSSION

First, the original centrifugal blower is experimentally tested. Fig.7 shows the original blower total pressure and efficiency compared to volume flow rate. Q_1 = 420 m^3/h state to the volume flow rate at best efficiency point (BEP) and Q_0 = 296.17 m^3/h states to the volume flow rate at highest total pressure point. Q_{min} = 0 m^3/h states to the minimum volume flow rate and Q_{max} = 665.14 m^3/h states to the maximum volume flow rate. The blower total pressure and volume flow rate are measured and the curve is plotted.

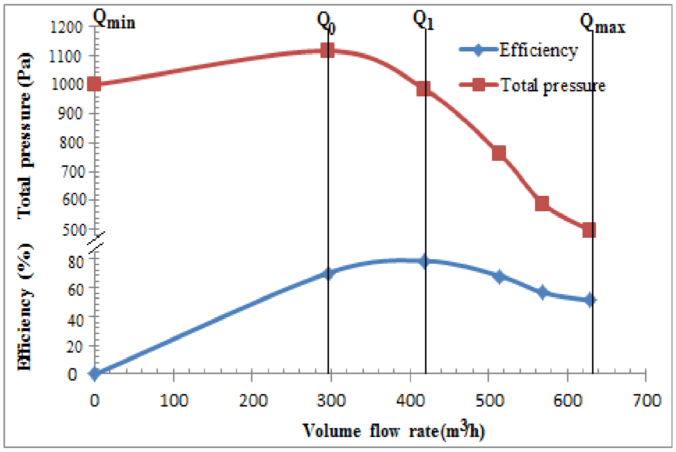

Fig.7. Original blower performance curve.

The blower total efficiency is calculated by using following equation,

$$\eta_{\text{Total}} = \frac{2.725\times10^{-3}\times Q\times P_t}{P_{sh}}$$

Where, P_t the blower total pressure and Q are the blower volume flow rate. The total efficiency is expressed as the ratio of the total power transfer to the fluid from the rotor to the actual power applied to the blower. The modifications of volute tongue clearance have affected on the total pressure, efficiency and shaft power of centrifugal blower. For different volute tongue clearances, the increasing volume flow rate the variation in total pressure was shown in Fig.8. As volume flow rate is greater than Q_0 = 296.17 m^3/h the total pressure is increased when the volute tongue clearance is decreased. For different volute tongue clearances, the increasing volume flow rate the variation in efficiency is shown in Fig.9. As volume flow rate is greater than Q_1 = 420 m^3/h the efficiency is increased when the volute tongue clearance is decreased. For different volute tongue clearances, the increasing volume flow rate the variation in shaft power is shown in Fig.10. At maximum volume flow rate Q_{max} = 665.14 m^3/h the shaft power is slightly increased when the volute tongue clearance is decreased.

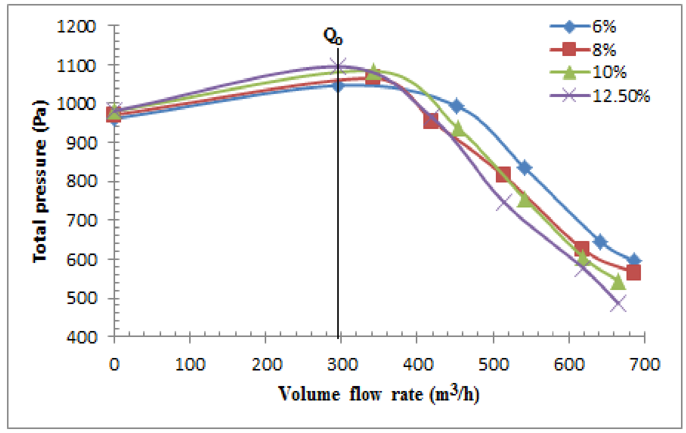

Fig.8. Effect of volume flow rate on total pressure.

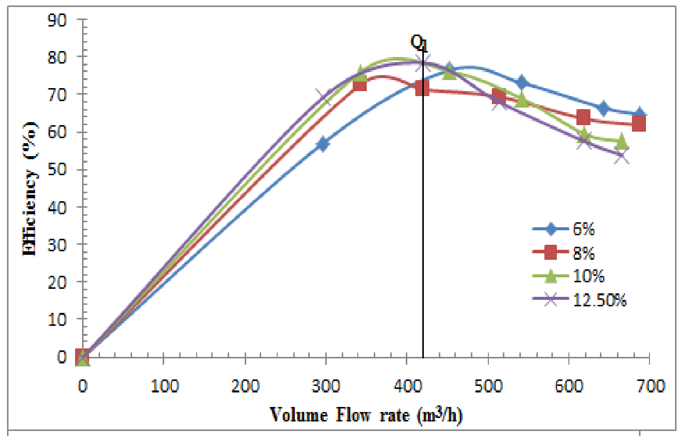

Fig.9. Effect of volume flow rate on efficiency.

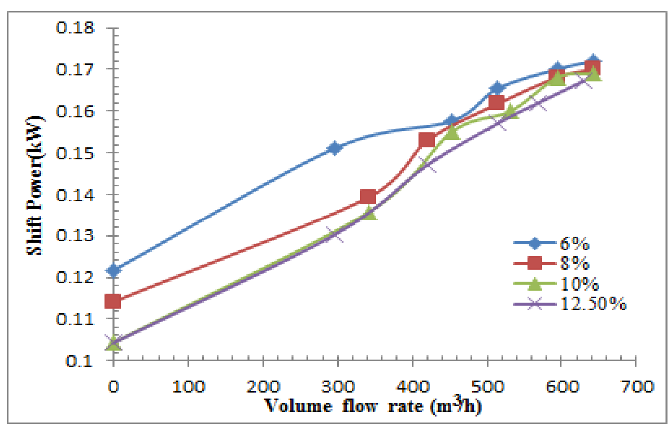

Fig.10. Effect of volume flow rate on shaft power.

Table III summarized the experimental results of the effect of volute tongues clearance on performance parameters like total pressure, shaft power and efficiency at full discharge condition. Table IV summarized the numerical results of the effect of volute tongues clearance on the total pressure at full discharge condition.

TABLE III

EFFECT OF VOLUTE TONGUES CLEARANCE ON PERFORMANCE PARAMETERS BY EXPERIMENTAL

Volute tongues clearance t_c/d (%)	Total pressure (Pa)	Efficiency (%)	Shaft power(kW)
12.5	486.634	53.774	0.1672
10	544.445	57.497	0.1690
8	564.065	61.850	0.1700
6	593.495	64.705	0.1720

TABLE IV
EFFECT OF VOLUTE TONGUES CLEARANCE ON TOTAL PRESSURE BY NUMERICALLY

Volute tongues clearance t_c/d (%)	Total pressure (Pa)
12.5	444.045
10	512.709
8	530.427
6	556.698

Comparing an experimental and numerical result from Table III and IV, it is found that the deviation of the total pressure of original and modified A_1, A_2, A_3 blowers are 42.5897 Pa, 31.735 Pa, 33.638 Pa and 36.797 Pa respectively. Fig. 11 shows the experimental and numerical results of the effect of variation of volute tongue clearance on the total pressure at full discharged condition. The numerical results are compared with the experimental results to confirm the validity.

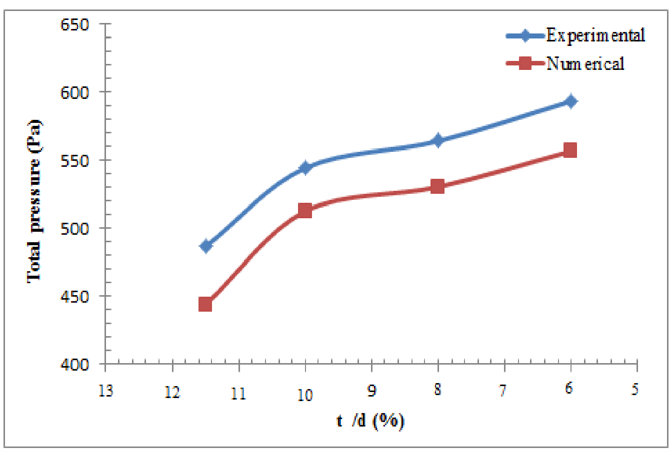

Fig.11. Effect of volute tongue clearance on total pressure.

VI. CONCLUSION

The present paper experimentally and numerically studies the effect of modification of volute tongues clearance of centrifugal blower on its performance. Experiments of the original blower and modified blowers are conducted and the results are compared.

1. As the volute tongues clearance decreases from 12.5% to 6% of impeller diameter, the total pressure is increased by 21.95% at full discharge condition.
2. As the volute tongues clearance decreases from 12.5% to 6% of impeller diameter, the efficiency is increased by 20.32% at full discharge condition.
3. The shaft power required to drive the impeller is little increases by 2.8%.when the volute tongues clearance of blower is decreased.

The modified volute tongues clearances, increases the total pressure and efficiency of blower because the reverse airflow at the region near blower volute tongue are effectively reduced. The numerical and experimental results shows that the efficiency and a total pressure of modified A_1, A_2 and A_3 blowers are higher than that of the original blower at a full discharge condition.

Appendix

P	Total pressure (Pa)
t_c	Volute tongue clearance (mm)
d	Impeller diameter (mm)
P_{sh}	Shaft power (kW)
η	Efficiency (%)
Q	Volume flow rate (m^3/h)
Q_0	Volume flow rate at BEP (m^3/h)
Q_{max}	Maximum volume flow rate (m^3/h)
Q_{min}	Minimum volume flow rate (m^3/h)
Q_1	Volume flow rate at highest total pressure point (m^3/h)
A_1	6% Volute tongue clearance blower
A_2	8% Volute tongue clearance blower
A_3	10% Volute tongue clearance blower
Z	Impeller blade number
BEP	Best efficiency point

References

[1] Fan handbook, Selection, Application and Design. Frank P. Bleier. (1998). Chapter no.7. pp. 7.1-7.45.

[2] Sheam-Chyun Lin, Ming-Lun Tsai. (2012, Dec).An integrated performance analysis for a backward-inclined centrifugal fan. *Computers & fluids*. 56.pp. 24–38.

[3] Qi Datong, Mao Yijun, Liu Xiaoliang, Yuan Minjian.(2009,April).Experimental study on the noise reduction of an industrial forward curved blade centrifugal fan. *Applied acoustic*. pp. 1042-1050.

[4] R. Dong, S. Chu, J. Katz (1997).Effect of modification of tongue and impeller geometry on unsteady flow, pressure fluctuations, and noise in a centrifugal pump. *The international gas turbine institute*. Vol. 119.

[5] Yew-Wah Wong, Weng-Kong Chan, and Wei Hu. (2007,Dec) Effects of tongue position and base circle diameter on the performance of a centrifugal blood pump. *International center for artificial organs and transplantation*. Vol.31. pp.639-645.

[6] Chen-Kang Huang, Mu-En Hsieh.(2009,May). Performance Analysis and Optimized Design of Backward-Curved Airfoil

Centrifugal Blowers. *American Society of Heating, Refrigerating and Air-Conditioning Engineers.* Volume 1., Number 3.

[7] O. P. Singh, Rakesh Khilwani, T. Sreenivasulu, M. Kannan.(2011, May). Parametric study of centrifugal fan performance: experiments and numerical simulation. *International Journal of Advances in Engineering & Technology.* Vol. 1,Issue 2,pp.33-50

[8] Sandra Velarde-suarez, Rafael ballsteros-Tajadura, Carlos Santo aria-Morros. (2006). Reduction of aerodynamic tonal noise of a forward curved centrifugal fan by modification of the volute tongue geometry. *Applied acoustics.* pp. 225- 232.

[9] W. Neise. (1975). Noise reduction in centrifugal fan: A literature survey. *Journal of sound and vibration*. pp. 375-405.

[10] W. Neise. (1982, May). Review of noise reduction methods for centrifugal fans. *Journal of engineering for industry*. Vol.104. pp. 151-160.

[11] William A. Smith, James K. O'Malley.(1974). Reducing blade passage noise in centrifugal fans. *American society of heating, refrigeration, and air conditioning engineers transactions*. Part II. Vol. 80. pp. 45-51.