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

Design of 5 kW Radial Type Centrifugal Blower (Casing)

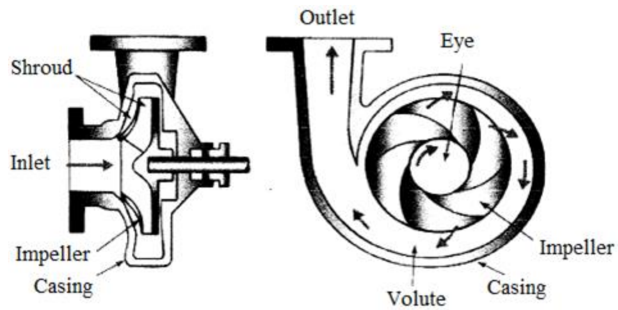
Aye Mya Thandar[1], Phyoe Min Than[2], Nang Sein Mya[3], Cho Cho Khaing[4]
[1, 4] Department of Mechanical Engineering, Technological University (Magway)
[2] Department of Mechanical Engineering, Technological University (Hmawbi)
[3] Department of Mechanical Engineering, Technological University (Myeik)

***Abstract-* There exist few design methodologies for centrifugal blowers and fans in the literature. However unified design methodology particularly for radial tipped centrifugal blowers and fans is not easily traceable in the literature available. In this research paper is attempted to design a single stage radial type centrifugal blower volute casing design for used in required industrial area. In this thesis, the bower volute casing is designed to provide low volume, high pressure air for cooling, ventilating and exhaust system that handle dust, materials or corrosive fumes. This centrifugal blower can produce 20 in of w.g (0.7396 psi) and 2l2l ft^3/min (60 m^3/min; of air at 60℉ and design speed 3450 rpm. In this research, the blower is designed air horsepower is 7 a.hp (5.2 kW) and the brake horsepower is 10 b.hp (7.5 kW). After choosing the design data from catalogue, the results of impeller inlet and outlet dimensions and vane angle could be obtained. This volute casing using Tabular Integration Method.**

***Indexed Terms-* Centrifugal blowers, fans, Design, Volute, power, Tabular Integration.**

I. INTRODUCTION

Air or gas enter the impeller axially through the inlet nozzle which provides light acceleration to the air before its entry to the impeller. The action of the impeller swings the gas from a smaller to a larger radius and delivers the gas at a high pressure and velocity to the casing. The centrifugal energy also contributes to the stage pressure rise. The flow from the impeller blades is collected by a spirally shaped casing known as scroll or volute. It delivers the air to the exit of the blower.



II. FLOW OUTLET FROM IMPELLER

The flow direction of the liquid at the outlet of the impeller can be:

1) Radial (perpendicular to inlet flow direction)
2) Mixed
3) Axial (parallel to inlet flow direction).

The flow outlet is determined by an important parameter called as the specific speed of the pump. As the specific speed of a pump design increases, it becomes necessary to change the construction of the impeller from a radial type to an axial type. Generally, it can be said that for low specific speeds (low flows and high heads) radial impellers are used whereas for high specific speeds (high flows and low heads) axial (propeller) impellers are used. The shape of impellers according to their specific speed as shown in Fig. 2.

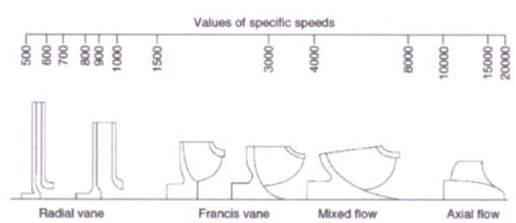


Fig. 2. Shape of Impellers According to their Specific Speed

III. DIFFUSERS AND VOLUTES

Static pressure is recovered from the kinetic energy of the flow at the impeller exit by diffusing the flow in a vane less or vaned diffuser. The spiral casing as a collector of flow from the impeller or the diffuser is an essential part of the centrifugal blower.

The provision of a vaned diffuser in blower can give a slightly higher efficiency than a blower with only a volute casing, for a majority of centrifugal blower, the higher cost and size that result by employing a diffuser outweigh its advantages. Therefore most of the single stage centrifugal blower impellers directly into the volute casing. Some static pressure recovery can also occur in a volute casing.

There is a small vaneless space between the impeller exit and the volute base circle. Volutes can be designed for constants pressures or constant average velocity. The cross-section of the volute passage may be square, rectangular, circular or trapezoidal. The fabrication of a rectangular volute from sheet metal is simple; other shapes can be cast. [4]

IV. DESIGN PROCEDURE OF RADIAL TYPE CENTRIFUGAL BLOWER (IMPELLER)

This research work is based on an industrial requirement for Twin city Blower Company, Aerovent. Radial blades have lower unit blade stress for a given diameter and rotational speed hence lighter in weight. There is equal energy conversion in impeller and diffuser so it gives high-pressure ratio with good efficiency [9, 20]. Looking to these realization and facts radial blade fan is selected for this study. The input parameters for the design of radial tipped centrifugal blower are as following,

Flow Discharge Q = 2121 ft^3/min
Static Suction Pressure = 14.7 psi (absolute)
Static Delivery Pressure = 0.7396 psi (gauge)
Static Pressure Gradient ΔPs = 981.2 Pa
Speed of impeller rotation N = 3450 rpm
Air Density = 1.165 kg/m^3
Outlet Blade Angle $\beta_2 = 90°$
Suction Temperature T_s= 60°F= (60 + 460) = 520°R
Atmospheric Pressure $P_{atm} = 1.01325 \times 10^5$ Pa
Atmospheric Temperature T_{atm} = 30°C = 303° K

These parameters are kept identical for each design methodology prescribed here in.

V. DESIGN OF VOLUTE CASING

For majority of centrifugal blowers, the higher cost and size that resulted by employing diffuser, overweigh its advantage. So most of single stage centrifugal impeller discharges directly into the volute casing, where some static pressure recovery can also occur.

Two most widely used methods of volute design areas;
1. Free vortex design.
2. Constant mean velocity design.

The purpose of the volutes as outline as outlined previously is to convert the velocity head of the gas leaving the impeller as efficiently as possible.

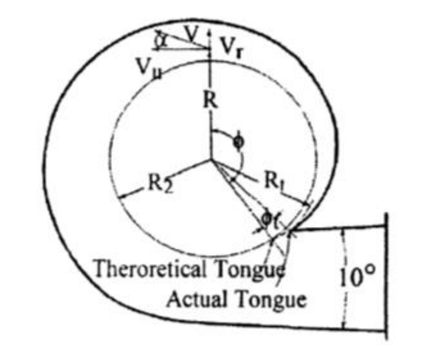


Figure 3. Elevation of Volute

The gas in the volute has very nearly the spiral flow in RVR = C = a constant, where C is determined from the relationship R2Vu2 = C for a given stage.

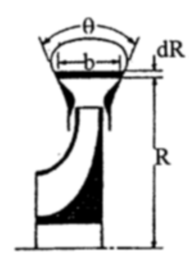


Fig.4. Section through Volute

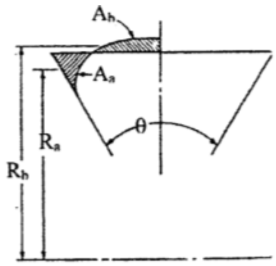


Fig.5. Volute Passage Cross Section

It may be assumed that the flow from the impeller is uniform about its periphery, so the flow past any section of the volute is Ø/360 of the total, where *e* is the angle in degrees measured from the theoretical tongue of the volute as shown in Fig. 3.

In determining the cross-sectional area of the volute at any point, the problem consists in finding the area of the section that will pass the volume Ø/360 with a velocity V_u=C/R it should be noted that the volute of Q used is that of delivered flow. It does not include the leakage flow which has now split off from the total impeller flow and returned to the suction through the wearing rings.

If friction is neglected, the flow through the differential section shown in Fig. 4. is
$dQ_{\Phi} = dAV_u = bdRV_u$

But V_u = C/R , here dQ_{Φ} = bdRC/R, and the total flow past the section becomes

$$Q\emptyset = \int_{R2}^{R\emptyset} dQ = C\int_{R2}^{R\emptyset} b\frac{dR}{R}$$

Where, R_{Φ} is the outer radius of a section at Φ from the theoretical tongue.

Substituting for dQ_{Φ} the term ΦQ /360 there results

$$\emptyset = \frac{360\,C}{Q}\int_{R2}^{R\emptyset} b\frac{dR}{R} = \frac{360\,R_2V_{u2}}{Q}\int_{R2}^{R\emptyset} b\frac{dR}{R}$$

After the shape of the side walls of the volute has been decided upon, the integral can best be solved by tabular integration as illustrated in the next section.

The shape of the volute is generally similar to that shown on Fig.4. The maximum total angle θ between the sides is usually about 60. If made greater the water is unable to follow the sides and turbulence and inefficiency result. Volutes with smaller 0 angles and larger radii give better results, but then the casing diameter and weight of the pump are increased unduly. If the discharge angle from the impeller α_2 is small, a larger angle between the sides may be used since the flow is then more nearly tangential. In order to simplify the calculations it may be assumed that the top of the volute is parallel to the shaft axis. After the areas have been found on this basis, the top wall of the volute may be made of any desired shape by substituting equivalent area for the original ones. Since RV_w is a constant and V_w =dQ/A at each section, it may be seen that dQR/A must be constant also, Hence $R_a dQ/A_a = R_b Q/A_b$ or $A_a/R_a = A_b/R_b$ and $A_b = A_aR_b/R_a$ where R_a and R_b are the distance from the shaft axis to the respective centers of gravity of the corresponding area as shown in Fig. 5.

These corrections may, in most cases, be neglected since the ratio R_b/R_a is usually very near unity and inherent casting inaccuracies destroy extreme refinements in calculation. It should also be remembered that the above calculation are based upon frictionless flow. Since the actual velocity is lower, the areas must be arbitrarily increased to care for this. This arbitrary increase in area may be much greater than the correction for the change of shape of passage.

To avoid shock losses the tongue angle should be made the same as the absolute outlet angle α_2 of the water leaving the impeller. The radius R_t at which the tongue starts should be 5 to 10 percent greater than the outside radius of the impeller to avoid turbulence and noisiness and to give the velocities of the water leaving the impeller a chance to equalize before coming into contact with the tongue.

The zero point of the volute or the point from which the angle Φ is measured may be found by assuming that the flow follows a logarithmic spiral. The equation of a logarithmic spiral is:
$R = R_2 e^{\tan\alpha_2\emptyset}$

Where, Ø is the angle measured in radians
α_2 is the constant angle of the spiral or the angle at which the water leaves the impeller
e is the base of natural logarithms=2.718

$\text{Log}_{10}\text{R}=\text{Log}_{10}\text{R}_2+\tan\alpha_2' \frac{\pi\emptyset°}{180} Log_{10}718$

Hence,

$$\emptyset_t = \frac{132 Log_{10}\frac{R}{R_2}}{\tan\alpha_2}$$

For the tongue radius $R=R_t$ and

$$\emptyset_t = \frac{132 Log_{10}\frac{R_t}{R_2}}{\tan\alpha_2{}'}$$

In the passage between the volute and discharge flange further conversion of velocity into pressure may take place, especially with high head pumps by making it divergent. In order to avoid turbulence the total divergence in this passage should not exceed 10°. They velocity here should never be greater than the minimum velocity the volute, otherwise some of the pressure energy would be reconverted to velocity. [2]

Table 1. Result Data of Impeller

Name	Symbol	Result
Shaft Diameter	D_s	1.5 in
Hub Diameter	D_H	2.5 in
Eye diameter	D_0	8 in
Eye velocity	V_0	100 ft/sec
Flow through eye	Q_0	35.5 ft^3/sec
Vane inlet diameter	D_1	8.25 in
Velocity at vane inlet	V_1	100 ft/sec
Impeller inlet width	b_1	2.5 in
Inlet vane angle	β_1	38°
Inlet vane thickness	ε_1	0.85 in
Number of vanes	z	15
Inlet tip speed	U_1	124.191 ft/sec
Outside dia. of impeller	D_2	18 in
Impeller outlet tip speed	U_2	270.962 ft/sec
Tangential component of outlet vel;	V_{u2}	214.212 ft/sec
Impeller outlet width	b_2	1.5 in
Outlet vane thickness	ε_2	0.93 in
Vane outlet angle	β_2	90°
Absolute vel;	V'_2	223.284 ft/sec
Absolute outlet angle:	α_2'	16°

VI. TABULAR INTEGRATION METHOD FOR CASING DESIGN

The preliminary shape of the volute section will be taken as a trapezoid with the side walls at 30° angle with the radial lines (θ= 60°) and a base width of 2.5 in at the impeller outlet D_2. This base width is found by adding to the outlet width b_2 twice the shroud width and twice the axial clearance. The width of the volute at any point may be scaled from a layout or calculated from the equation $b = b_3 + 2x\tan(\theta/2)$, where x is the distance between any radius R and the impeller rim radius R_2.

A. The base width:

$b_3 = b_2 + (2 \times \text{shroud width}) + (2 \times \text{axial clearance})$

$= 1.5 + (2 \times 1/4) + (2 \times 1/4) = 2.5$ in

If angle between sides is θ = 60°, the width of volute at any point will become

$b = b_3 + 2 \times\tan(60/2)$

$b = 2.5+2\times\tan30°$

The tongue radius R_t is made about 10 percent more than the tip radius R_2.

$R_t = 1.1 \times 9 = 9.9$ in

The tongue angle $\emptyset_t$ may be found from following eqn;

$$\emptyset_t = \frac{132\text{Log}_{10}\frac{R}{R_2}}{\tan\alpha_2}$$

For the tongue radius $R=R_t$ and

$$\emptyset_t = \frac{132\text{Log}_{10}\frac{R_t}{R_2}}{\tan\alpha_2{}'}$$

$\text{Log}_{10}\frac{R_t}{R_2} = \log_{10}1.1=0.0414$

$\alpha_2{}'=16°$

$\tan\alpha_2' = 16°$

The volute is designed by determining the angle $\emptyset°$ measured from an assumed radial line by tabular integration.

$$\emptyset° = \frac{360\, R_2 V\prime_{u2}}{Q_2}\int_{R2}^{R\emptyset} b\,\frac{dR}{R_{ave}} = 130.558\int_{R2}^{R\emptyset} b\,\frac{dR}{R_{ave}}$$

$$R_{ave} = R + \frac{\Delta R}{2}$$

$$b_{ave} = b_3 + 2(R_{ave} - R_2)\tan\frac{60}{2}$$

$$\Delta A = b_{ave} \times \Delta R$$

$$Q\emptyset = \frac{Q\emptyset}{360}$$

$$V_{ave} = \frac{Q\emptyset}{A\emptyset}$$

The discharge flange diameter is based upon air 223.284 ft/sec and the flow of discharge Q_2 is 35.4 ft^3/sec. The diameter of discharge flange will be

$$D = \sqrt{\frac{4Q}{\pi V}} = \sqrt{\frac{4 \times 35.4 \times 144}{\pi \times 233.284}} = 6 \text{ in}$$

Table 2. Result for Geometry of Volute Casing

R	ΔR	R_{ave}	b(in)	$\frac{b\Delta R}{R_{ave}}$	ΔØ	Ø	ΔA	AØ	QØ	V_{ave} (ft/s)
9			2.5			0		0		
	1	9.5	3.07	0.33	44		3.07			
10						44		3.07	4.22	197.9
	1	10.5	4.23	0.4	54		4.23			
11						98		7.3	9.34	184.2
	1	11.5	5.38	0.46	62		5.38			
12						160		12.68	15.24	173.1
	1	12.5	6.54	0.52	70		6.54			
13						230		19.22	21.92	164.2
	1	13.5	7.69	0.56	76		7.69			
14						306		26.91	29.11	155.7
	1	14.5	8.85	0.61	82		8.85			
15						388		35.76	36.81	148.5

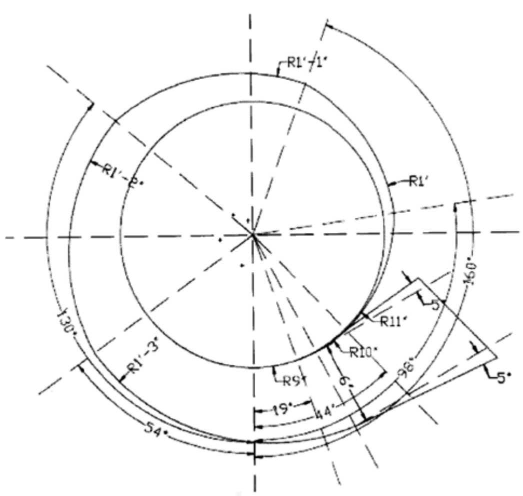


Fig.7. Geometry of Volute Casing Using Tabular Integration Method

VII. CONCLUSION

In this paper, the design consideration of different volute casing at best efficiency point (BEP) for commercial aspect. In designing blower volute, it is therefore important to utilize all exact dimensions as like volute throat area, volute mean diameter, volute circumferential length and minimizing casing losses. Blower can vary in size from a few feet to tens of feet in diameter, depending on their application. Centrifugal blowers are very useful in many industries, factories and many engineering fields. Although the centrifugal blowers are existing machines in many respective fields, we must be understood in its detail design, construction, function, application and basic theoretical knowledge. The design blower can be developed a pressure of 15.4396 psi (106.452 kPa) and deliver 2l2t ft3/min (60 m3/mim) of air at 3450rpm. The designed impeller has B25 in (209.55 mm) inlet diameter, 18 in (457.2 mm) outlet diameter, 380 inlet vane angle and 90□ outlet vane angle. The number of vanes is 15 blades. And then, the inlet width and outlet width are 2.5in (63.5mm) and1.5in (38.1mm) respectively. The diameter of discharge flange is 6 in (152.4mm). So, this research would be very useful and applicable for the vane shape of designing blower that is widely used in industries.

REFERENCES

[1] Pump Handbook, Igor J.Karassik. Third Edition, 2001.

[2] Centrifugal Blower Ebara Corporation 1998'

[3] Fan Performance and Selection' ASHARE Handbook' 1992'

[4] Centrifugal Pumps and Blowers. AUSTIN H. CHURCH. 1972

[5] Energy Efficiency Guide for Industry in Asia' 2006'

[6] Jang, C. M., and Yang, S. H., 2009, "Performance Characteristics on Turbo Blowers Connected in Serial and Controlled by Inverter," The 10th Asian International Conference on Fluid Machnery, Malaysia, pp. 685–695.

[7] Lin, S. C., and Huang, C. L., 2002, "An Integrated Experimental and Numerical Study of Forward-Curved Centrifugal Fan," Experimental Thermal and Fluid Science, Vol. 26, No. 5, pp. 421–434.

[8] Yu, Z., Li, S., He, W., Wang, W., Huang, K., and Zhu, Z., 2005, "Numerical Simulation of Flow Field for a Whole Centrifugal Fan and Analysis of the Effects of Blade Inlet Angle and Impeller Gap," HVAC&R Research, Vol. 11, No. 2, pp. 263–283.

[9] Zhang, J.; Chu, W.; Zhang, H.; Wu, Y.; Dong, X. Numerical and experimental investigations of the unsteady aerodynamics and aero-acoustics characteristics of a backward curved blade centrifugal fan. Appl. Acoust. 2016, 110, 256–267.

[10] Jiang, Y.Y.; Yoshimura, S.; Imai, R.; Katsura, H.; Yoshida, T.; Kato, C. Quantitative evaluation of flow induced structural vibration and noise in turbo machinery by full-scale weakly coupled simulation. J. Fluids Struct. 2007, 23, 531–544.

[11] Jie Jina Ying Fan, Wei Han, Jiaxin Hu, 2012. "Design and Analysis on Hydraulic Model of The Ultra - low Specific-speed Centrifugal Pump", International Conference on Advances in Computational Modeling and Simulation, 31: 110-114.

[12] Li Chunx, Wang Song Ling and JiaYakui, 2009. "The performance of a centrifugal fan with enlarged impeller", Journal of Sound and Vibration, 4(8): 87.

[13] Shojaeefard, M.H., M. Tahani, M.B. Ehghaghi, M.A. Fallahian, M. Beglari, 2012. "Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid", Computers & Fluids, 60: 61-70.

[14] Pham Ngoc Son, E. Jaewon Kim, Y. Ahn, 2011. "Effects of bell mouth geometries on the flow rate of centrifugal blowers", Journal of Mechanical Science and Technology, 25(9): 2267-2276.

[15] Singh, O.P., 2012. "Parametric study of centrifugal fan performance", International Journal of Advances in Engineering and Technology, 3(2): 33.

[16] Sun-Sheng Yang, Shahram Derakhshan, Fan-Yu Kong, 2012. "Theoretical, numerical and experimental prediction of pump as turbine performance", Renewable Energy, 48: 507-513.

[17] Taguchi, G., 1992. "Taguchi Methods - Research and Development", ASI Press, Dearborn, MI.