2 Chapter REVIEW OF LITERATURE

2.0 GENERAL

Blower has been widely used and one of the well established fluid handling machinery. Survey is done to get information about the research material available for Volute casing of Blower. To get the detailed design, Experimental and Computational Fluid Dynamics investigation to predict the flow in a blower casing, which produces a complex three dimensional phenomenon involving turbulence, secondary flows and unsteadiness. Also, Pressure fluctuation interacts with the Volute casing. This pressure distribution inside the casing is measured with various techniques. The distributions of pressure and efficiency obtained by numerical simulation can be a good agreement with those determine experimentally. Analysis of flow in casing is predicted by using CFD and which is very useful tool in visualizing the flow at various section in volute. It is describe that performance characteristic of blower as a function of the shape, expansion diameter and width of the volute casing. All these can be used to optimize the design of volute casing of blower. Selection of material for the construction of volute casing can lead to cost reduction. To get the complete knowledge of all above factors for the construction of efficient volute casing large survey is required. Literature survey has been divided into the different areas of work done by the researchers as, in the field of geometric parameters of the casing, Experimental work done over the casing and recent work done as per the latest technology and software available to analyze the flow in Volute casing of blower.

2.1 GEOMETRIC PARAMETERS

A two-dimensional model of the flow in a vaneless diffuser is represented in figure: 2.1. It represents the spiral vortex path that, traces a streamline from the impeller outlet at point 2 to the casing at point 3.

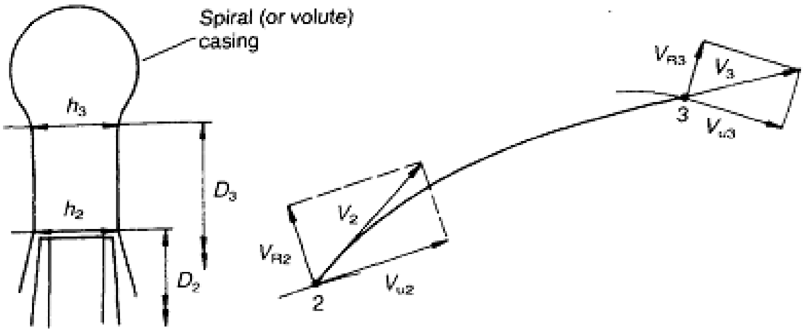

Figure: 2.1 shows entry of Velocity in Casing through diffuser.

These theoretical approaches are guideline for experimental studies.

2.1.1 VOLUTE AREA

The total flow of the impeller discharge passes through the volute throat area; only part of the impeller capacity passes through any other section. Worster [6] studied the casing and its effect on pump performance and proposed that the most important parameter was the volute throat area. According to Stepanoff [7] if the volute areas are too small in comparison with the optimum values, the peak efficiency will decrease slightly and more towards a lower capacity. When volute area is too large the peak efficiency may increase but will move towards a higher capacity. Efficiency at partial capacities will then be lower.

Worster argued that the best efficiency or design match between the impeller and casing occurs when the impeller and volute characteristics are cross as indicated in figure: 2.2. He based on his analysis on the simple casing cross-section and maintained that although the outlet angle and impeller diameter do determine performance, the correct matching of impeller and casing is a decisive factor in design process. Figure:2.3 illustrate his point & figure: 2.4 is his design plot that allows the selection of the correct volute area once the impeller is designed.

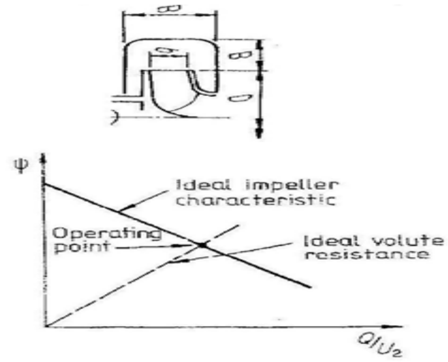

Figure: 2.2 Illustration of the approach to impeller and casing matching propose.

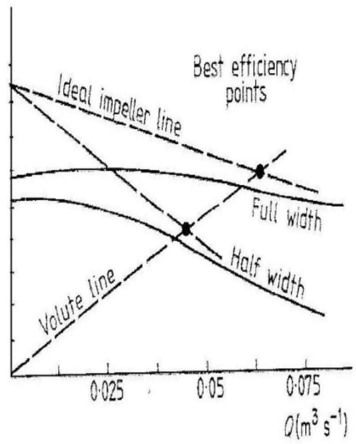

Figure: 2.3 Illustration of volute area change on pump performance

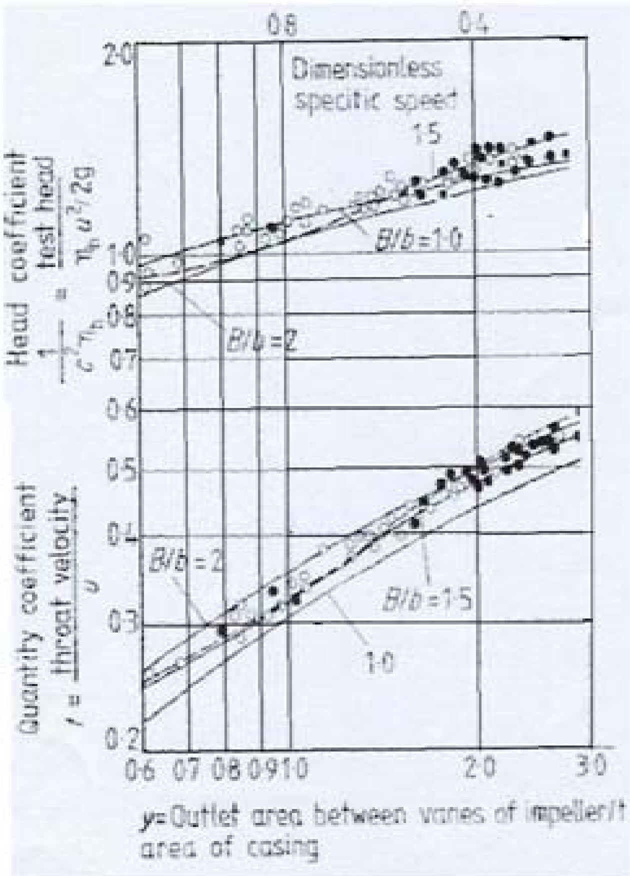

Figure: 2.4 Design plot proposed by Wroster

2.1.2 VOLUTE ANGLE & TONGUE:

To avoid shock and separation loss at the volute tongue the volute angle α_v is made to correspond to the direction of the absolute velocity vector at the impeller discharge flow angle. Usually there is an ample gap provided between the impeller and the volute tongue for the flow to adjust itself for a minimum loss. At higher specific speeds the volute angle α_v and the length and shape of the tongue become more important.

Stepanoff has found that removal of a portion of the tongue reduced the efficiency from 85 to 81% while restoring the tongue to the original shape brought the efficiency back to 85%.

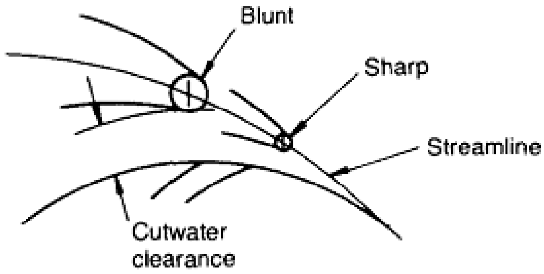

Figure: 2.5 Different shape of Tongue.

The shape of the tongue is also important according to information published by Turton [3]. Depending upon casing the tongue may be blunt or sharp but must be aligned along the streamline as shown in the figure: 2.5 blunt noses allows for tolerance of flow misdirection and thus an accommodation for flow changes from design point.

2.1.3 VOLUTE WIDTH

To establish the volute width the continuity equation of the several volute elements are connected by knowing the values of d_4 and A_v. b_4 is a function of the volute angle, higher volute angle require narrow volute i.e. taller volute section. Stepanoff [7] says that a volute with width exceeding the impeller width is more efficient than a volute having an equal width. Values of b_4 vary from 1.25 b_2 to 2 b_2 (where b_2 is impeller outlet width), the high values applying to low impeller or the same impeller with a reduced diameter without any appreciable effect on the efficiency. To get the detail flow analysis inside the casing, flow measurement is done by various researchers.

2.2 EXPERIMENTAL MEASUREMENT OF FLOW

The recirculation of flow effects in the spaces between impeller and casing resulting radial loads over the bearings supporting the shaft tend to be large.[1] In papers by Turton and Goss(1982-1983) it is shown that the fluctuating forces may be of the same size as the steady load and this factors as well as the fluctuating nature of the load effects gives rise to vibration and bearing and shaft failure in some cases. The reason for the fluctuating components is felt to be the varying velocity distribution leaving the impeller and interacting with the casing.

Abidogun has carried out detailed experimental investigation to document the effects of vaneless diffuser diameter and width ratios on the fundamental characteristics of flow perturbation in the vaneless diffuser of a centrifugal blower [9]. In addition, a benchmark set of experimental data will be provided to aid numerical investigation of instabilities in the vaneless diffuser flow field in particular and the entire compression system in general. The current data showed that a decrease of the diffuser width, at constant impeller angular speed, resulted in a decrease of the critical flow coefficient. The data further revealed that a decrease in the diffuser diameter ratio resulted in an increase in the frequency of rotating stall. Variation of the diffuser width does not have any appreciable effect on the frequency of rotating stall. The effects of flow coefficient on stall characteristics are also reported.

Saad a. Ahmed and Mohamed A Gadalla has carried out an experimental investigation to delay the onset of the rotating stall in the radial diffuser of a centrifugal blower [10]. The method involved reducing the flow area by 50% at the diffuser exit using throttle rings attached to either diffuser shroud wall, or the diffuser hub wall. Simultaneous attachments of the throttle rings to both the diffuser's shroud and the hub walls were also made. The following blockage ratios were used: 25% from both walls, 50% from the diffuser shroud and 0% from the diffuser hub and vice versa. The preliminary results suggest that the onset of the flow instability in the diffuser (stall) could be delayed (i.e., lower flow coefficient) when throttle rings were attached to the diffuser walls to reduce its exit flow area. The results also confirmed that the throttle rings could be an effective method to control/delay the stall in the vaneless radial diffuser.

M.Govardhan and G.Venkateswarlu have carried experiments on investigate the effect of impeller geometry and tongue shape on the cross flow fans [11]. Each impeller was tested with two tongue shapes. Flow survey was carried out for each impeller and tongue shape at two flow co-efficient. The total pressure developed by impellers in each case is found to be maximum at a circumferential position of around 270˚. Experiment shows flow parameters like total pressure, static pressure in general do not vary along the span except near the shrouds, indicating that the flow is predominantly two dimensional in cross flow fans.

According to the work done for proper design of centrifugal fans by D.M.Sharma[12] it is essential to estimate the slip factor correctly. He has used several co-relations for estimating the slip factor which concludes that for a given specified fan the value of slip factor is constant & is dependent of impeller geometry only. But on the basis of fewer historical evidences, approach & experience this statement has been challenged and found to be partially correct and factual evidences were proved which suggested that the slip factor not only depends on the geometry of the impeller but also on the specific speed and flow rate. Moreover the value does not remain constant at any location for the exit of air through impeller blades. Also the experimental value is found to be considerably smaller (3 -12%) than the predicted values based on various correlations due to occurrence of possible discrepancies.

Whitfield and Baines [13] have given the design of the blower. This design is restricted to impeller only. Earlier study was done by Stanitz [14] considering only one dimensional flow in the casing, and it has found that to get the detail analysis, three dimensional analyses is required. Experimental Investigation by Hagelstein [15] predicted that the flow in a blower produces a complex three dimensional phenomenon. For the three dimensional measurement and visualization of the flow phenomenon Eckardt [16] had worked to get the instantaneous characteristic of flow. Bryer [17] have developed the flow measurement with help of pressure probe. But Pressure recovery occurs in the volute and it may be effected at off design conditions is not predicted. Arttu REUNANEN [18] has found that, the volute causes a non-uniform pressure distribution which leads to a radial force on the impeller. Also, Pressure fluctuation interacts with the Volute casing and gives rise to dynamic effects over the mechanical parts, which are one of the most important sources of vibration and hydraulic noise. How to minimize the effect is not analyzed and what will be at off-design condition.

The distribution of pressure around the periphery of Blower due to Volute can be design in following ways.

(a) Constant Angular Momentum Design.

This type of design is also known as Free Vortex flow[19]. A centrifugal impeller of diameter'd' and a point lying outside the impeller with a radius 'r' one obtained a whirl velocity C_θ from the condition that the rotation must remain constant i.e

$$r * C_\theta = \text{Constant} \qquad \text{-------------------------------}(1.15)$$

(b) Constant Velocity Distribution Design.

It has been established experimentally that Volutes with a constant average velocity for all section results in the best efficiency [5]. This means that volute area increases in proportion to their angular displacement from the tongue where the area is Zero.

$$C = R_c * C' \qquad \text{-------------------------------------}\ (1.16)$$

Where – C is Average Volute Velocity.

R_c is Volute Velocity Distribution factor.

C' is absolute velocity at the impeller discharge.

As per both design size of the casing differs for same inlet conditions. In constant velocity method one has depends upon Volute velocity distribution factor. There can be various cross-section of Volute passage such as Rectangular, Trapezoidal, Circular, Inner Volute, axial volute, subdivided volute, with number of discharge points [20]. As per requirement the cross-section can be selected. Rectangular cross-section is simple in fabrication, Trapezoidal results turbulence in flow and also leads to heavier volute than other methods [21].

2.3 RECENT DEVELOPMENT:

Tarek Meakhail and Seung O. Park have done the velocity measurement data in the interaction region between the impeller and vaned diffuser and the results of numerical flow simulation of the whole machine (impeller, vaned diffuser and volute) of a single stage centrifugal fan [8]. Two dimensional instantaneous velocity measurements are done using particle image velocimetry (PIV). Numerical simulation of impeller-diffuser-volute interaction is performed using CFX-Tascflow commercial code. The simulation results show that the separated flow regime near the diffuser hub extends to the volute. Comparison between the unsteady computation and those of measurement indicates that the rotor/stator model employed in the simulation predicts essential characteristics of unsteady flow in the centrifugal blower. However, quantitative agreement remains rather poor. Below in the table some outcome of the review is summarized.

Researcher	Work done and Conclusions	Discussion
Fukutomi J, Itabashi A and Senoo Y 2006	1) Tested centrifugal blower with volute casing with large tongue clearance and concluded that at design point. 2) The ratio of circulating flow rate to total flow rate at the volute end is about 45%, and the pressure recovery in the volute is about 45%. As the remaining 55% of the pressure recovery in the casing is recovered in the Section between volute and exit duct. 3) Casing acts as a high-performance diffuser. The centrifugal force produced by the secondary flow and boundary layer suction are the main causes of large pressure recovery	1) Working at design point is discussed. 2) This type of application and requirement are not specified. 3) Development of boundary layer is theoretical discussed, which is quite different from actual. 4) This is basic of all the casing.
Michio Kitadume 2007	1) The multi-blade fan, many factors must be considered in its practical design because the flow generated in the fan is quite complicated with three-dimensionality and unsteadiness. 2) The fundamental fan performance is primarily determined by the impeller of the fan, and is also affected by the scroll casing. 3) The theoretical estimation of the effect of the casing on the performance has not been well established. In order to estimate the casing effect on fan performance, detailed three-dimensional (3D) flow analysis in the casing is necessary. 4) Stereoscopic PIV (SPIV) is one of the useful techniques for experimental analysis of 3D flow fields. There are some difficulties in practical application of SPIV for flow analysis in fluid machinery with complicated geometry, but the results obtained provide useful information for understanding the 3D flow field. 5) In this report, experimental investigation of the flow in the	1) Effect near the tongue region is not predicted. 2) Interference of wall is not considered. 3) Case is only for multi-blade fan, with very small region. 4) With various conditions is to be predicted.

	scroll casing has been carried out using PIV and SPIV under the premise of downsizing automobile air conditioner fans.	
P Epple, B Karic, C Ilic, S Becker, F Durst and A Delgado 2008	1) A combined analytical and numerical method for designing of BC blower was presented suited to design high-efficiency radial impellers. 2) The torque-speed characteristic of the motor without the need of a previous impeller series or knowledge of experimental data, allowing optimizing efficiency in the design stage. 3) The blade shapes were computed using an inverse method. An efficiency increase of 13% and a maximum flow rate increase of 11%. 4) The design point was also met. It was concluded that β_2 less than 16° will not increase efficiency further, rather induce more losses.	1) Work is regarding theoretical method. 2) Blade fabrication at lower angle is too complicated and separation is more. 3) Casing parameters are not discusses.
Tan J J and Luo T F 2008	1) A new method was presented to calculate the Euler work of centrifugal impellers of blowers considering the presence of an inlet flow prewhirl in the case without the inlet guide vane. 2) Stodola's approach dealing with the slip velocity at the impeller outlet was applied to the impeller inlet. 3) A new formula for Euler work calculation was deduced to evaluate the effect of the inlet prewhirl. 4) The new formula had been applied to 33 industrial centrifugal fans and the calculated results had been compared with the experimental data of these fans. 5) The comparison showed that the new formula was more accurate in most cases than the original formula without consideration of the inlet flow prewhirl, and the accuracy had been improved by more than 10 per cent on average.	1) It is a theoretical work. Experimental results are to be required. 2) This theory is for impeller correction, but how it will affect the casing performance as a whole is not reflected. 3) It is comparison of the same formula, but with other formula how it will react is not cleared.
Huang, Chen-	1) A backward curved centrifugal blower was numerically	1) Design factors

Kang 2009	simulated and analyzed. 2) The results from numerical simulations and measurements were compared to verify the validity of numerical simulation. 3) The effects of blade angle, blade number, tongue length, and scroll contour were numerically studied. 4) Some favored parameter changes were determined and utilized to redesign one of the blowers. 5) The optimized model indeed exhibits a better value of cost/performance.	are not considered. 2) Role of Casing is not considered. 3) Optimization method is not cleared 4) Validity of numerical simulation with respect to casing is not discussed.
Choon-Man Jang 2010	1) In this paper, the performance characteristics of a turbo blower as a function of the shape of the volute casing: expansion diameter and width of the volute casing. 2) The turbo blower considered in the present study is mainly used in a refuse collection system. 3) The flow characteristics inside the turbo blower were analyzed by a three-dimensional Navier-Stokes solver and compared with experimental results. 4) The distributions of pressure and efficiency obtained by numerical simulation were in good agreement with those determined experimentally. 5) Throughout the numerical simulation of the turbo blower, the blower performance was enhanced by decreasing the local losses in the blade passage and the outlet flow. 6) The efficiency and pressure for the design flow condition were enhanced by about 3% and 2%, respectively, compared to the efficiency and pressure of the reference blower. 7) Detailed flow analysis was performed using the results of the numerical simulation.	1) Numerical simulation at tongue region is not shown. 2) Effect of position of Tongue angle is not discussed. 3) Losses in blade are discussed not in casing. 4) Effect of casing on the efficiency is not discussed.
Choon-Man Jang 2011	1) Optimal operation of the turbo blowers having an inlet vane has been studied to understand the blowers operating performance. 2) Analyze three-dimensional flow field in the turbo blowers	1) It contains only the inlet effect. 2) Turbo blower in series is studied.

	serially connected, general analysis code, CFX is introduce.	
Morris, G.K. 2012	1) Variable speed drives are typically air-cooled and fall under recent international standards for improved fan efficiency. 2) This paper details the modeling, design optimization, and experimental verification approaches used to optimize blower housing designs for variable speed drives. 3) The design of the blower housing is just as important as the blower selection. By modifying the housing dimensions, the shape and quantity of flow exiting the housing can be controlled.	1) Design is made as per Variable speed drives, but in industry requirement can be change. 2) How the impeller effect can be match with casing is not cleared. 3) As the speed will change load will be varied, so what will be the pressure effect is not studied.
Dr.S.A.Channiwala 2012	1) Numerical analysis of the single stage centrifugal blower is carried out for different flow coefficient. 2) Analyze of the 3-D flow field, fluid domain is created and simulation is done with the CFD software code ANSYS CFX. 3) Three dimensional Navier-Stokes equations are used to analyze the flow. Standard k-ε turbulence model and unstructured grid is adapted to solve the Navier-Stokes equations. 4) Results of numerical analysis suggest that the pressure coefficient at an outlet of fluid domain is continuously decreased with increase in flow coefficient. 5) The detail flow analysis is also carried out for the practical case of flow coefficient. 6) The flow is analyzed at different angular and axial positions inside the volute.	1) Only numerical analysis is carried out. 2) Simulation with frozen rotor is not carried out.
A.T. Oyelami 2012	1) The level of convergence of results obtained from the failure analyses equally indicate the reliability of the results that the impeller will not fail in service.	1) Impeller analysis is carried out. 2) Casing details are not

	2) The data obtained can serve as a tool in selecting blower design type for different applications. 3) The qualitative and quantitative analyses have also shown that backward curved vanes have the best performance cum efficiency in industrial blowers applications	mentioned.
Choon-Man Jang and KA-Ram Choi 2012	1) In his work describes the optimization of the impeller having splitters for a turbo blower. 2) Two design variables, chord of splitter and pitch of splitter, are introduced to enhance the blower performance.	1) Role of splitters are studied at design condition. 2) Working at off-design is not cleared, as it will create problem in casing.

2.4 SCOPE AND OBJECTIVES OF THE PRESENT WORK

After review of all the above literature it has been found that still much work is required to get the detail study of flow and design of medium range blower casing. Many factors must be considered in its practical design because the flow generated in the blower is quite complicated with three dimensionality and unsteadiness. The fundamental blower performance is primarily determined by the impeller of the blower and is also affected by the casing. However, the theoretically estimation of the effect of the casing on the performance has not been well established. In order to estimate the casing effect on the Blower performance, detailed three-dimensional flow analysis in the casing is necessary.

1) Experimental work is required to gain the knowledge about the three dimensional flow field at various angular, radial, axial locations and near the tongue region in the Volute casing and make the analysis regarding the losses that occurs due to the design of casing in medium range blower with backward curved impeller for which information is not available as per survey. Major work is in the field of compressor but not in the medium range Blower.
2) Experiment is to be carried out with three dimensional calibrated probe connected to a standard manometer for measurement of pressure upto 500mm to 700mm of water and flow direction. To traverse the probe at different angle and position, versatile traversing mechanism is required. The range of movement should be as per the height of the casing. This will lead to accurate and promising tool for measurement.
3) For validation of the readings uncertainty analysis is to be carried out. This is not seen in any Literature survey of the blower.
4) By considering free vortex or constant velocity method for design of casing does not depends on the selection of pressure requirement, as a criterion for design of casing. Therefore,

Design of the casing should be identical and uniform for all the type of impeller. This is required to make uniform design. In Constant velocity design method as per equation 1.16, design is based on constant value R_c which is taken from experimental reference chart. So there should be common design for the casing which satisfies the end user as per pressure requirement.

5) Design of the blower casing should be optimized to get various range of efficiency with considering various parameters. There are many constraints to be satisfied in optimization, which is not done until now. This is very important now days, as to save power and cost.
6) As per the industrial need consumption of the power is to be reduced and blower should be operated as per requirement in the system. So, by application of variable speed drive in the system the performance of the casing getting affected is to be studied.
7) As there is no information regarding mismatching of impeller in the casing. What should be the range for fitting the impeller in the casing, if the casing is designed as per impeller having outlet blade 40^0 and we want to operate the blower by fitting the impeller with 70^0. So with this condition what will be the effect on the system? This information can be economical to an industry.
8) There is no information regarding use of Volute Casing as suction to another Blower system, which can be used for application involving filtration and collect the smoke from the surrounding.
9) To validate theoretically the flow in the casing of the blower, CFD analysis is required. It is observed in high pressure compressor but not in medium range blower. This can save the experimental cost and time. The results of CFD are more explanatory. So, in medium range blower casing it will help in

analysis for change in cross section and selection of material for fabrication.

All above short comings can help in better design and understanding of the flow in volute casing of the blower. So, problem formation is divided in to three main parts, experimental work, design part and CFD work. To investigate the flow in volute casing of the blower, experimental setup is planned by using existing motor having 7.5hp (5.5kw) at rotational speed of 2900r.p.m. and impeller of backward straight blade with inlet blade angle 20^0 and outlet blade angle 48^0 with tangential direction having thickness of the blade 5mm, inner and outer diameter 300mm and 425mm and number of blades 12. With these existing impeller casing is to be designed by both method of "Free Vortex” and "Constant Mean Velocity" by considering the volute width 260mm, twice the impeller width. Flow in this casing is to be measured with help of five-hole probe made of s.s material having tubes of 1mm diameter, outer tube of 6.5mm and length of 800mm.

For mismatching of impeller in the above casing, impeller is to be designed having forward curved blade with inlet blade angle 20^0 , outlet blade angle 120^0, blade thickness 5mm, inner diameter 300mm, outer diameter 425mm and number of blades 12 and measurement is to be done with the help of calibrated five-hole probe. To check the validity of the Experimental results Uncertainty analysis is to be done.

After the experimental performance CFD analysis is to be planned with the all above conditions and also static and dynamic analysis should be done in ANSYS.

After the experimental and CFD analysis, design will be optimized with objective of increasing the efficiency of the system. Considering the various Independent variables such as β_2, Z, d_1, b_1 and b_2 and satisfying the constraints such as $V_{eye} < 22$ m/s, $d_1 / d_2 = 0.5$ to 0.8, $(d_1/d_2) / (1.194*\varphi^{1/3}) = 1$ to 1.1, $b_1/d_1 = 0.208$ to 0.46, $b_2 < b_1$ and divergence angle < 12°, $\beta_1 = 15°$ to 35°, $V_{m1}/V_{m2} = 1.25$ to 1.6,

W_1/W_2 > 1.05. Effect of this optimization is to be analyzed by considering two methods such as Genetic Algorithm method and combined heuristic method.

Results of this optimization will be validate by performing the experimental setup with the optimized design of the casing having backward inclined impeller with inlet angle of 30.41^0 and outlet angle of 70.62^0 with tangential direction, thickness of 2mm, inner diameter of 100mm, outer diameter of 260mm having number of blade 10 and volute will be fabricated from 2mm thickness ms sheet and width twice the impeller width. This defined problem of Investigation of flow inside the casing will leads to the improvement of Blower performance.

2.5 SUMMARY

The present Literature survey highlights research work in multiple dimensions with respect to Blower casing. Literature survey is done as per effect of geometry of casing such as volute casing shape and width. In which only theoretical work is done, but to verify in medium range blower experiment is required. Later survey is done for the experimental work done by the various researchers. It is observed that experimental work is done in compressor only, not in medium range Blower. Recent development in the field of flow analysis in blower shows that various researchers have used latest technology of flow measurement such as PIV and CFD analysis in compressor, but not in Blower. After the Literature review it is found that, still there is a lack of information available in the medium size Blower. Literature survey has enhanced the problem formation of investigation of the flow in Blower volute casing. In the scope of investigation of flow in Blower volute casing is divided in three parts Experimental work, design and optimization and CFD analysis. Detail experimental work is required, to validate the optimization and CFD analysis regarding the flow. Many factors must be considered in its practical design because the flow generated in the blower is quite complicated with three dimensionality and unsteadiness. The fundamental blower performance is primarily determined by the impeller of the blower and is also affected by the casing. However, the theoretically estimation of the effect of the casing on the performance has not been well established. In order to estimate the casing effect on the Blower performance, detailed design analysis in the casing is necessary. Survey has encouraged to developed uniform design methodology for blower casing by considering various constraint. Optimal design has become a matter of interest in the Blower design with the rapid development of a computational power. Optimized design with various constraints is build up for overcoming the uniform methodology for the design of volute casing.