INTRODUCTION

1.1 Centrifugal fan:

The centrifugal fan uses the centrifugal power generated from the rotation of impellers to increase the pressure of air/gases. When the impellers rotate, the gas near the impellers 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 ducts. After the gas is thrown-off, the gas pressure in the middle region of the impellers decreases. Main parts of a centrifugal fan are Fan Housing, Impellers, Inlet and outlet ducts, Drive Shaft, Drive mechanism. [11]

It has a fan wheel composed of a number of fan blades, or ribs, mounted around a hub. As shown in Figure 1, the hub turns on a driveshaft that passes through the fan housing. The gas enters from the side of the fan wheel, turns 90 degrees and accelerates due to centrifugal force as it flows over the fan blades and exits the fan housing.

Fig (1) Schematic diagram of Centrifugal fan

1.2 1 Types of blade:

1. Backward-curved blades, $\beta_2<90^\circ$
2. Radial Blades $\beta_2=0$
3. Forward-Curved Blades, $\beta_2<90^\circ$

Figure (2) various types of impeller vane

Fans are one of the types of turbo machinery which are used to move air continuously with in slight increase in static pressure. Fans are widely used in industrial and commercial applications from shop ventilation to material handling, boiler applications to some of the vehicle cooling systems, power and energy source, environment, chemical engineering, etc. With expansion of its use in industry, the centrifugal fan is developed towards the direction of high efficiency and energy saving. Selection of fan system depends on various conditions such as airflow rate, temperature of air, pressures, airstream properties, etc. The fan is always analysed by its performance curves which are defined as the plot of developed pressure and power required over a range of fan generated air flow. The best method of improving the performance of the fan is to understand exactly the three dimensional turbulent flow in the centrifugal fan. The performance curves of a turbo machine can be obtained by theory, computation, and by series of experiments. The experimental analysis is difficult, costly and time consuming. To evaluate predicted performance of theoretical design, various computational methods are available. They offer optimum design solutions without actual fabrication or making prototypes which save time and expenditure. The CFD part is used for improvement the results of Static Pressure generated at the entry to the impeller, static efficiency. The CFD optimization also helped to improve the flow pattern through the centrifugal fan system.

The numerical procedure thus developed requires three functional input parameters volume flow rate (Q, cfm), static pressure, and fan speed (rpm) which gives the output parameters such as number of blades, static efficiency, total efficiency, velocities at entry and exit of the impeller and the entry and exit angles of the blades. Also, the numerical design gave the details of volute casing. The most important and initial step in numerical simulation is geometry definition and grid generation of computational domain. This process includes selection of grid types, grid refinements and defining correct boundary conditions.

Earlier investigation has attempted to identify the general flow pattern inside the fan. In early 1980s,[1] Raj and Swim (1981) studies the flow at the exit of a forward curved centrifugal fan rotor by using the smoke technique and hot wire probes. Centrifugal fan involves flow through rotating impeller and vaned or
vaneless stationary diffuser. Meakhail and Park [2] have studied the impeller-diffuser-volute casing interaction in fan experimentally and validated it numerically. They have used steady analysis results as an initial parameter for unsteady analysis later on. Steady-state interaction between volute and impeller is studied by Fahua Gu, Engeda [3], using frozen rotor model and explained the role of volute in reduced efficiency at off-design conditions. Power loss occurs due to fluid drag on the reverse surface of the impeller back plate as rightly pointed out by W.C. Osborne [4].

N. Vibhakar et al. [5] experimented on a Backward Curved Radial Tipped Blade Centrifugal Fan. In this study, fan geometry is obtained as per unified design methodology and CFD analysis carried out in this work is to understand the volute-impeller interaction at design and off-design conditions under varying mass flow rates, rotational speeds and number of blades. Steady, realizable k-ε model with MRF approach is used to evaluate the flow behaviour inside centrifugal fan by using ANSYS software. Performance curves are obtained under the Rotational speed of impeller constant and varying, Number of impeller blades as 12, 16 and 24 and Volume flow rate from 0.1 to 0.5 m3/sec. In Figure 2.1 Maximum total efficiency 45.89%, which occurs at 0.25 flow coefficient.

**Figure 3: Performance curves of Efficiency Vs. flow coefficient for constant rotational velocity N=2800 rpm with 12, 16 and 24 numbers of blades with design point.**

**Figure 4: Performance curves of Efficiency Vs. flow coefficient for 2500,2650 and 2800 rpm keeping No. of blades Z=16 constant**

Figure 2 show the effect of varying rotational speed while 16 numbers of blades kept constant show that power coefficient and efficiency are almost unaffected near design low coefficient and has different values at lower and higher flow coefficients. Hence all the quantities varying with flow coefficient are increased as the number of blades increases. The streamlines clearly shows the rotating effect of blades on flow, within and outside of impeller zone, which is imposed using MRF approach. The numerical analysis shows that, for efficient energy transfer i.e. to achieve optimum performance in centrifugal fan designed by unified methodology number of blades should increase.

O. P. Singh et al. [6] discussed about in this paper, effect of geometric parameters of a centrifugal fan with backward- and forward-curved blades has been investigated. The parameters considered in this study are number of blades, outlet angle and diameter ratio. And also Effect on the vehicles mileage due to the use of forward and backward fan is also discussed. Figure 5 show that (a) As the flow coefficient increases the difference between the pressure coefficient of 12 and 18 blades increases and opposite happens when flow coefficient decreases. E.g. at lowest flow coefficient pressure coefficient of fan 2 is only 12% higher than fan 1. However at highest flow coefficient the difference is 30%, (b) Fan efficiency and power coefficient shows similar trends, (c) In the same limit of flow coefficients, efficiency varies from 1.75% to 8.0% and, (d) power coefficient varies between 9% to 34%. To an experimental engineer it means that if they evaluate the fan performance in the actual system they will be able to see only 12%, 1.75% and 9% difference in performance with 12 and 18 blades fan. The results show that increase in the number of blades increases the flow coefficient accompanied by increase in power coefficient. However, difference in the performance (efficiency, flow and power coefficient) tends to decrease at higher pressure coefficient. The results suggest that fan with different blades would show same performance under high-pressure coefficient. Increase in the number of blades increases the flow coefficient and efficiency due to better flow guidance and reduced losses. The efficiency of the fan first increases and then decreases with diameter ratio. The best efficiency of the fan was observed to be at diameter ratio of 0.5.

**Figure 5: Outlet angle on performance characteristics of fan angles at 4000 rpm.**

Xiaomin Liu et al. [7] In this paper centrifugal impeller firstly by solving the Navier-strokes with the spalart-allmaras turbulence model and the performance curves are then obtained. And effect between the inlet duct and impeller inlet on the performance of the centrifugal fan is studies numerically. According to the calculated result, the linkage profile between the inlet duct and impeller inlet is redesigned to improve the performance of the fan. It seen that total pressure and the efficiency increase compared with original centrifugal fan. Then the flow in an inlet duct, an impeller, a diffuser and a volute is analyzed compared with an impeller, the efficiency of the centrifugal fan drop by about 3-4%
because of flow loss in the volute. Effects of a straight shroud with different inclined angle, the performance of the centrifugal fan increased and otherwise decreases.

**Lin and Tsai et al.** [8] Study about an integrated performance analysis for a backward-inclined centrifugal fan and performance evaluation of fan design under different operating conditions. Performance analysis for a BI centrifugal fan is carried out through a combined experimental and numerical approach. An 80 mm-diameter backward-inclined centrifugal fan is chosen to serve as the research subject for demonstration purposes. Numerical results are utilized to perform detailed flow visualization, torque calculation, efficiency estimation, and noise analysis. The results indicate that the fan performance curve and the sound pressure level (SPL) spectrum of the experiment agree with those of numerical simulations and flow visualization at each operating point, having verified the successful enhancement of fan performance via numerical calculation.

**Lin and Huang et al.** [9] Conducted preliminary experimental and numerical study of FC centrifugal fan. A small centrifugal fan is successfully designed for the thermal task of cooling laptop computers by utilizing an integrated scheme, which consists of fan design, mock-up manufacture, experimental verification, and numerical simulation. In this research, a cooling fan is designed and manufactured by the computer numerically controlled (CNC) machine to carry out the corresponding experimental verifications. By comparing the experimental and numerical results, a good agreement between them indicates a great potential to reduce expensive experimental work by using CFD tool.

**K. Vasudeva Karanth et al.** [10] Study about the Effect of Radial Gap on impeller-diffuser flow of a centrifugal Fan figure 3. The flow between the impeller exit and the diffuser entry (i.e., in the radial gap is generally considered to be complex). With the development of PIV and CFD tools such as moving mesh techniques and numerical methodology involving moving mesh technique is used in predicting the real flow behaviour, as exhibited when a target blade of the impeller is made to move past corresponding vane on the diffuser.

Result found that there is an optimum radial gap at which better dynamic and static heads are developed by the impeller blades as well as better energy conversion by diffuser vanes and maximum efficiency of the centrifugal fan as observed in the study.

**Fig. 6: Model of the centrifugal fan used in the analysis**

**Tahsin Engin et al.** In the present experimental study, three semi-open centrifugal fan impellers have been designed and fabricated using ceramic materials to provide high resistance to temperature. Experiments have been conducted to investigate the performance characteristics of these impellers and the deteriorations in their performance due to varying tip clearance. Factors have been determined to estimate the tip clearance losses. Experiments have been conducted on the gases with temperatures up to 1050 °C. The effect of impeller geometry shaft speed, gas temperature, and the tip clearance on the overall performance of the tested impellers have been studied experimentally. The closed and open forms of a typical fan impeller with main dimensions are shown in Fig.6, in which the absolute tip clearance is indicated by x. The typical design is done based on the required pressure rise and flow rate. In order to increase the fan internal efficiency, the following parameters are usually changed without reducing design pressure and flow rate: d1/d2.

**Fig. 7: Tip clearance in radial flow impellers: (a) Closed impeller and (b) semi-open impeller**

Results showed that the simple impeller geometries of ceramic materials were less sensitive to the varying tip clearance and the gas temperature has been found to have almost no influence on the performance degradation due to the tip leakage flow.

**MY COMMENTS:-**

From the above review it is conclude that the following scope of work.

- CFD analysis of centrifugal fan
- number of blade increases
- number of blade decreases
- blade inlet angle to be changed
- blade outlet angle to be changed
- CFD software results compare with the actual tested results and get maximum head

**CONCLUSIONS:**

In this paper, an investigation on the effect of centrifugal fan parameters on performance has been presented thorough experiments and a CFD simulation has been presented. Test results show that increase in the number of blades increases the flow coefficient accompanied by increase in power coefficient. Increase in the number of blades increases the flow coefficient and efficiency due to better flow guidance and reduced losses. The best efficiency of the fan was observed.

**REFERENCE:**

3. Fahua Gu and Abraham Engeda, A numerical investigation on the volute/impeller steady-state interaction due to circumferential distortion,


