(IJITR) INTERNATIONAL JOURNAL OF INNOVATIVE TECHNOLOGY AND RESEARCH
Volume No.5, Issue No.4, June – July 2017, 6915-6918.

The Effect of Impeller Blade Design on Centrifugal Blower – CFD Approach

GOLLA JYOTHI
M.Tech (Thermal Engineering) Student, Dept.of Mechanical Engineering, Sanketika Institute of Technology & Management, (Approved by AICTE and Affiliated to JNT University, Kakinada) Beside Cricket Stadium, P.M Palem, Visakhapatnam-530041

A. VINUTHA
Assistant professor, Dept.of Mechanical Engineering, Sanketika Institute of Technology & Management, (Approved by AICTE and Affiliated to JNT University, Kakinada) Beside Cricket Stadium, P.M Palem, Visakhapatnam-530041

Abstract: **A centrifugal blower is a mechanical device for moving air or other gases with a slight increase in pressure. While working with hot air, the heat absorbed by the impeller material should be dissipated effectively to avoid the accumulation of heat at one place, that may deform the part. The rate of dissipation depends on the heat transfer rate and the thickness of the blade material. The present work deals with the determination and analysis of heat transfer rate of blade made of Aluminum alloy, composite materials Graphite and Carbon Fiber with blade thickness 1mm, 2mm, 3mm. This is done using ANSYS software in thermal analysis. CREO parametric software is used for modeling the centrifugal blower. The performance of Blower depends on the change in static pressure for different input velocities of air. In CFD analysis change in pressure, outlet velocity, heat transfer coefficient of air is determined at different input velocities 14m/sec, 16m/sec, 18m/sec, 20m/sec and 22m/s.**

Keywords: **Centrifugal Blower; CREO; ANSYS; Composite Materais;**

I. INTRODUCTION

A centrifugal fan is a mechanical device for moving air or other gases. The terms "blower" and "fan" are frequently used as synonyms. These fans increase the speed and volume of an air stream with the rotating impellers.

Centrifugal fans are constant displacement devices or constant volume devices, meaning that, at a constant fan speed, a centrifugal fan moves a relatively constant volume of air rather than a constant mass. This means that the air velocity in a system is fixed even though the mass flow rate through the fan is not.

A centrifugal blower consists of an impeller which has blades fixed between the inner and outer diameters. The impeller can be mounted either directly on the shaft extension of the prime mover or separately on a shaft supported between two additional bearings.

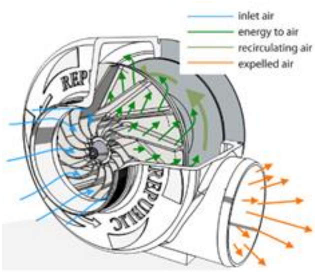

Fig.1 Centrifugal blower

Air or gas enters the impeller axially through the inlet nozzle which provides slight 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 flow from the impeller blades is collected by a spiral-shaped casing known as *volute casing* or *spiral casing.* The casing can further increase the static pressure of the air and it finally delivers the air to the exit of the blower.

The performance of a centrifugal blower mainly depends on the design of impeller blade. Impeller blades of aerofoil shape are good.

Centrifugal fans are by far the most prevalent type of fan used in the HVAC industry today. They are often cheaper than axial fans and simpler in construction. They are used in transporting gas or materials and in ventilation system for buildings. They are also well-suited for industrial processes and air pollution control systems.

II. LITERATURE SURVEY

Static and Dynamic Analysis of a Centrifugal Blower Using FEA in this project work this paper is used to study static and dynamic analysis of blower so as to reduce vibrations & impact. The present work aims at examining the choice of composites as an alternative to metal for better vibration control. Composites, known for their superior damping characteristics are more promising in vibration reduction compared to metals. The modeling of the blower was done by using solid modeling software, CATIA V5 R19. The blower is meshed with a three dimensional hex8 mesh is done using HYPERMESH 10. It is proposed to design a blower with composite material, analyze its strength and deformation using FEM software. In order to evaluate the effectiveness of composites and metal blower using FEA packaged (ANSYS). Modal analysis is performed on both Aluminums and composite blower to find out first 5 natural frequencies.

Et al[2]**Atre Pranav** and **Thundil Karuppa Raj R**

III. THEORETICAL ASPECTS

3.1 Construction

Main parts of a centrifugal blower are:

- Fan housing
- Impellers
- Inlet and outlet ducts
- Drive shaft

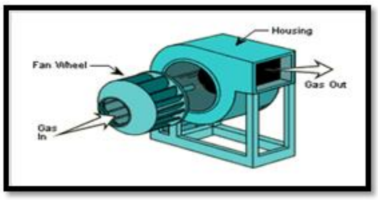

Fig. 2 Components of a centrifugal blower

3.2 Introduction to CREO

PTC CREO, formerly known as Pro/ENGINEER, is 3D modeling software used in mechanical engineering, design, manufacturing, and in CAD drafting service firms. It was one of the first 3D CAD modeling applications that used a rule-based parametric system. Using parameters, dimensions and features to capture the behavior of the product, it can optimize the development product as well as the design itself. It has many advantages like Optimized for model-based enterprises, Increased engineer productivity, Better enabled concept design, Increased engineering capabilities, Increased manufacturing capabilities, Better simulation, Design capabilities for additive manufacturing.

3.3 Introduction to CFD

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing, e.g. flight tests.

IV. CFD ANALYSIS OF CENTRIFUGAL BLOWER

→→Ansys → workbench→ select analysis system → fluid flow fluent → double click

→→Select geometry → right click → import geometry → select browse →open part → ok

→→ select mesh on work bench → right click →edit → select mesh on left side part tree → right click → generate mesh →

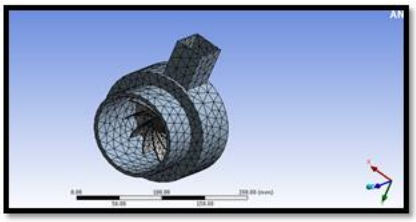

Fig 3. Meshed model

Select faces → right click → create named section → enter name → water inlet

Select faces → right click → create named section → enter name → water outlet

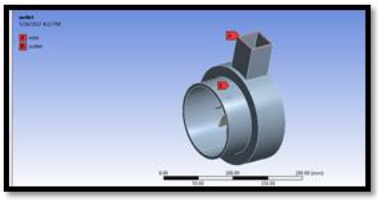

Fig. 4 Inlet and Outlet positions

Model → energy equation → on.

Viscous → edit → k- epsilon

Enhanced Wall Treatment → ok

Materials → new → create or edit → specify fluid material or specify properties → ok

Boundary conditions → select air inlet → Edit → Enter air Flow Rate → 2Kg/s and Inlet Temperature – Solution → Solution Initialization → Hybrid Initialization →done

Run calculations → no of iterations = 50 → calculate → calculation complete

→→ Results → graphics and animations → contours → setup

4.1 Case 1. Input Velocity of Air – 14 m/sec

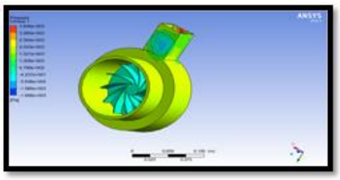

Fig. 5 Pressure Variation

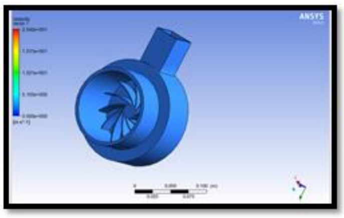

Fig. 6 Velocity Variation

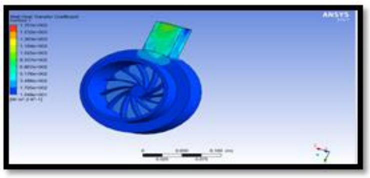

Fig.7 Heat Transfer Coefficient Variation

V. THERMAL ANALYSIS OF CENTRIFUGAL BLOWER

Open work bench 14.5 >

select **steady state thermal** in analysis systems > select geometry > right click on the geometry >

import geometry > select **IGES** file > open

The model is generated in CREO Parametric software to the dimensions and is imported.

Model >right click>edit>select generate mesh

Boundary conditions

Select steady state thermal >right click>insert>

Select steady state thermal >right click>insert>select heat flux

Select steady state thermal >right click>solve

Solution>right click on solution>insert>select temperature

The model is divided into fine triangular elements called as meshing.

Material properties

Aluminum Alloy

Thermal conductivity = 15.1W/m K

Specific heat =356J/Kg K

Density = 4120 Kg/m^3

Graphite

Thermal conductivity = 24.0 W/m K

Specific heat =707.7 J/kg ^{o}C

Density = 2.25g/cc = 2250 kg/m^3

Carbon fiber

Thermal conductivity = 900 W/m K

Specific heat =1135.0 J/kg ^{o}C

Density = 1.7 g/cm^3 = 1700 kg/m^3

5.1 Case 2. Thickness of Blade – 1mm

Material of Blade - Aluminum

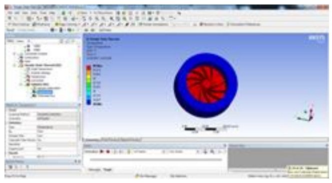

Fig.10 Temperature Variation

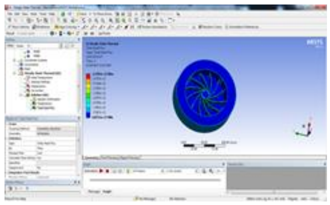

Fig.11 Heat Flux Variation

Table 1. THERMAL ANALYSIS RESULTS TABLE

Thickness of blade	Material of blade	Temperature of inlet air (°C)	Heat Flux (W/mm2)
1mm	Aluminum alloy	90	3.239e-12
	Graphite	90	4.9714e-12
	Carbon fiber	90	9.94e-12
2mm	Aluminum alloy	90	2.2335e-12
	Graphite	90	3.4196e-12
	Carbon fiber	90	6.8392e-12
3mm	Aluminum alloy	90	1.2904e-12
	Graphite	90	1.9767e-12
	Carbon fiber	90	3.9534e-12

Table 2. CFD ANALYSIS RESULTS TABLE

Inlet velocity (m/s)	Pressure (pa)	Heat transfer coefficient (w/m^{2}k)	Velocity (m/s)
14	2.802e+04	7.18e+03	1.180e+02
16	3.576e+04	8.33e+03	1.371e+02
18	4.542e+04	9.43e+03	1.546e+02
20	5.615e+04	10.50e+03	1.732e+02
22	6.196e+04	10.91e+03	2.680e+02

Table 3. COMPARISON OF HEAT TRANSFER COEFFICIENT ANALYSIS VALUES WITH THEORETICAL VALUES

Inlet velocity (m/s)	By CFD analysis (w/m^{2}k)	Theoretical value (w/m^{2}k)	% Deviation
14	7.186e+03	9276.22	19.85%
16	8.33e+03	10332.04	19.30%
18	9.432e+03	11352.94	16.92%
20	10.50e+03	12351.34	14.98%
22	10.91e+03	13329.95	18.15%

Table 4. PERFORMANCE OF BLOWER

Inlet velocity of air (m/sec)	Pressure (k Pa)	Efficiency of blower (%)
14	2.802e+01	13
16	3.576e+01	21.28
18	4.542e+01	36.36
20	5.615e+01	63
22	6.196e+01	84.67

VI. CONCLUSION

It is observed that the heat transfer rate of the blade decreases with increase in its thickness from 1mm to 3mm. And its value is more for blade made of Carbon Fiber material than that of Graphite material. Aluminum has the least value of heat transfer rate. Hence Carbon Fiber blade of 1mm thickness possess good heat dissipation quality.

By observing the CFD analysis the pressure change, outlet velocity and heat transfer coefficient values of the air increases by increasing the inlet velocity. Analytical values are compared with that of theoretical values and was found that the percentage deviation of the value of heat transfer rate of air is reasonable.

Performance of the Blower is determined for the flow values obtained at different input velocities from CFD analysis and it was found that Efficiency increases with increase in input velocity.

VII. REFERENCES

[1] S.Rajendran and Dr.K.Purushothaman, “Analysis of a centrifugal pump impeller using ANSYS-CFX,” International Journal of Engineering Research & Technology, Vol. 1, Issue 3, 2012.

[2] S R Shah, S V Jain and V J Lakhera, “CFD based flow analysis of centrifugal pump,” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[3] P.Usha Shri ans C.Syamsundar, “computational analysis on performance of a centrifugal pump impeller,” Proceedings of the 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, IIT Madras, Chennai, 2010.

[4] E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, “Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle,” The Open Mechanical Engineering Journal, no 2, 75-83, 2008.

[5] Marco Antonio Rodrigues Cunh and Helcio Francisco Villa Nova, “Cavitation modelling of a centrifugal pump impeller,” 22nd International Congress of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil, 2013.

AUTHOR’s PROFILE

GOLLA JYOTHI, M. Tech (Thermal Engineering) Student, Dept.of Mechanical Engineering, Sanketika Institute of Technology & Management, (Approved by AICTE and Affiliated to JNT University, Kakinada) Beside Cricket Stadium, P.M Palem, Visakhapatnam-530041

A. VINUTHA, Assistant professor, Dept.of Mechanical Engineering, Sanketika Institute of Technology & Management, (Approved by AICTE and Affiliated to JNT University, Kakinada) Beside Cricket Stadium, P.M Palem, Visakhapatnam-530041

