A Parametric Study of the Aerodynamic Characteristics of a Centrifugal Fan with Splitter Vanes

Author
Sohaib Hassan Khan
00000206104

Supervisor
Dr. Tariq Talha

DEPARTMENT OF MECHANICAL ENGINEERING
COLLEGE OF ELECTRICAL & MECHANICAL ENGINEERING
NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY
ISLAMABAD

AUGUST, 2021

A Parametric Study of Aerodynamic Characteristics of a Centrifugal Fan with Splitter Vanes

Author
Sohaib Hassan Khan
00000206104

A thesis submitted in partial fulfillment of the requirements for the degree of

MS Mechanical Engineering

____
Thesis Supervisor
Dr. Tariq Talha

DEPARTMENT OF MECHANICAL ENGINEERING
COLLEGE OF ELECTRICAL & MECHANICAL ENGINEERING
NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY
ISLAMABAD

AUGUST, 2021

DEPARTMENT OF MECHANICAL ENGINEERING

COLLEGE OF ELECTRICAL & MECHANICAL ENGINEERING

NATIONAL UNIVERSITY

OF SCIENCES AND TECHNOLOGY

ISLAMABAD

AUGUST, 2021

Declaration

I certify that this research work titled *"A Parametric Study of Aerodynamic Characteristics of a Centrifugal Fan with Splitter Vanes"* is my own work. The work has not been presented elsewhere for assessment. The material that has been used from other sources has been properly acknowledged and cited.

Sohaib Hassan Khan
00000206104
MS-2017 Mechanical Engineering
College of Electrical & Mechanical Engineering
National University of Sciences and Technology, Islamabad

Language Correctness Certificate

This thesis has been read by an English expert and is free of typing, syntax, semantic, grammatical, and spelling mistakes. The thesis is also written according to the format provided by the university.

Sohaib Hassan Khan
00000206104
MS-2017 Mechanical Engineering
College of Electrical & Mechanical Engineering
National University of Sciences and Technology, Islamabad

Dr. Tariq Talha
Assistant Professor
Department of Mechanical Engineering
College of Electrical & Mechanical Engineering
National University of Sciences and Technology, Islamabad

Copyright Statement

Acknowledgements

I am immensely grateful to my thesis supervisor, Dr. Tariq Talha, for his support, continuous encouragement and constructive criticism that has enabled me to successfully complete my thesis. I am also thankful to him for his guidance and advice regarding research and career prospects during and after my MS studies.

I would also like to express my heartfelt gratitude to Dr. Faraz Ahmed Kunwar and Dr. Sajid Ullah Butt for providing invaluable guidance and feedback during this work, and for being on my thesis guidance and evaluation committee.

I would like to extend my gratitude to my colleague, Mr. Saleem Akhter, for several enlightening discussions. The support of the staff at the Supercomputing Research and Education Center in the Research Center for Modeling and Simulation, National University of Sciences and Technology, is also gratefully acknowledged, as is the kind permission for the use of the facilities at this center.

Dedicated to my loving parents for their invaluable support and cooperation

Abstract

A forced fan is one of the most vital assembly in an engine cooling system. The type of fan is backward- curved centrifugal fan. Its mechanically driven. Primary objective of a forced fan is to pass large volumes of fresh air in order to cool the engine.

Considerable research is done on the related topics in recent past. In most of the past papers, effects of splitter vanes are investigated for a variety of turbo-machines. Specially related to pumps. Research done on the effects of splitter vanes for fans, blowers and compressors is limited.

In this thesis, aerodynamic characteristics of a forced fan with implementation of splitter vanes is analyzed. A total of three design parameters pertaining to splitter vanes are studied in-depth. Effect of each design parameter is investigated for multiple dimensions. After performing series of simulations and experiments, the margin of improvement in fan efficiency is analyzed.

The overall increase in fan efficiency is found for the modified geometry. It's found that implementation of splitter vanes has significant effect on the fan efficiency. However, geometric parameters related to splitter vanes have negligible effect on splitter vanes.

It's recommended, to perform experimental validation of optimal geometry obtained by this study. In future, volute casing of the subject forced fan can be optimized aerodynamically. Since an engine cooling system is constituted by a number of assemblies. It's recommended to conduct the research in order to optimize performance of each component in a cooling system.

Table of Contents

List of Figures

List of Tables

CHAPTER ONE

1 Introduction

The research interests of scientists are driven by the virtue of industrial demand. Fans, Blowers, Compressors have huge market value and industrial applications. Notable areas of applications are power, energy, chemical processing, aviation and space flight. [1] Hence, topic remains under R&D by the researchers and scientists. Mostly researchers have focus on optimized design of rotating equipment, performance enhancement, noise reduction and weight reduction. Designers always seek to minimize the effort required in fan development and time required for design.

Traditional available tools for testing and optimization were insignificant prior to numerical tools. Velocity vector diagrams were made. Lab wind tunnel experiments were done. Experimental models and setups are expensive to execute and time consuming. Experimental models also had higher chances of errors and inaccuracies. It is by the virtue of readily available numerical tools that researchers can perform spontaneous simulations. Now a days, readily available CFD tools are capable enough to simulate turbo-machinery.

In this thesis, aerodynamics of a forced fan is analyzed. Each design parameter is studied, how it effects the overall performance of fan efficiency. This research is focused on three geometric design parameters.

A forced fan is one of the vital assemblies in an engine cooling system. Forced fan is of a centrifugal type and mechanically driven. It passes large volumes of fresh air through water radiators and oil coolers. This is done as fan converts the kinetic energy into pressure head. Thus, it lowers the operational temperature of an engine. Optimization of a fan will enhance overall efficiency of a cooling system.

Analyses of a fan requires deep insight of design and its parameters. and how each parameter will affect the fan performance. Impact of parameters on the overall performance of the fan is a primary focus of this research. This research comprises of six chapters. The most important chapters are chapter two, chapter five and chapter seven. Chapter two is focused on fan theory and operation. Chapter four is based on the numerical simulation setup. It tells step by step procedure to configure the software for analysis. Chapter five is based on the results and discussions. Critical argument is done in favor of best design point.

1.1 Background

A splitter re-aligns the fluid flow. [2]. Splitter vanes are partial blades in between the main blade. Length, height and angle of splitter vanes is varied subjected to design and application. Quite a lot of research has been done on splitter vanes in recent past. Applications of splitter vanes are mostly rotating machinery like compressors, pumps, blowers and fans. Scientists and researchers are always seeking efficient and performance enhancement designs. Optimization of design is primary objective where there is a design constraint. Major concerns of performance enhancement and optimization is to develop a cost-effective design, efficient equipment performance and energy conservative designs.

Splitter vanes are very well known for performance enhancement. As splitter

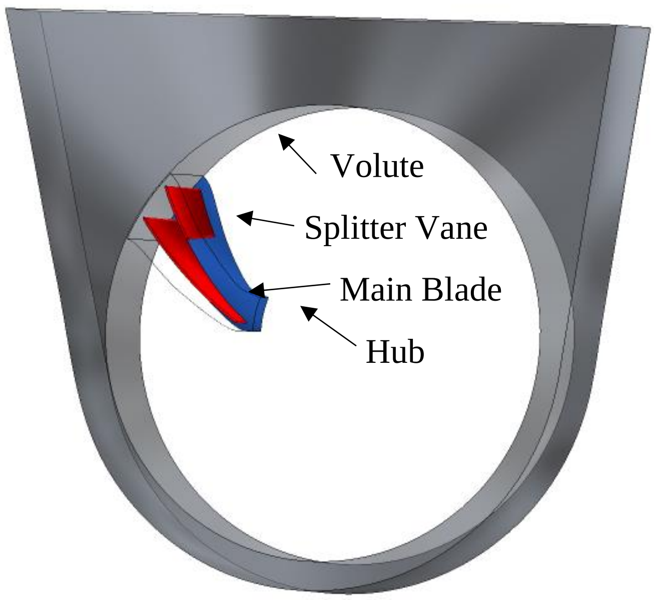


Figure 1.1: *Rendering of volute with casing*

vanes reduces the back flows and flow separations at trailing edge. This is done as splitter vanes allows more fluid to pass through the rotating impeller by guiding it.

Implementation of splitter vanes is popular amongst designers; as there is no inevitable design alteration. Performance is enhanced in strict compliance with the geometric constraints. Parameters like major diameter, height and blade numbers can remain same as that of reference geometry. With slight increase in weight, significant increase in pressure and efficiency can be obtained.

1.2 Literature Review

Different research papers, journals and articles directly/ indirectly related to the topic of our research were studied in depth. [3] Chen-Kaug Huang and Mu-En Hsieh numerically simulated the backward curved centrifugal blower. Experimental results were compared with numerical and the deviation of 4.8% and 15.1% was observed for pressure and efficiency respectively. Increased efficiency is obtained by optimizing the geometric parameters like number of blades, blade angle, tongue length and dimensions of scroll contour. 7.9% of static pressure and 1.5% improvement in efficiency is observed. [4] Effect of splitter vanes is analyzed on a centrifugal fan by Man-Woong Heo, Kyung-Hun Chu, Jin-Hyuk Kim and Kwang-Yong Kim. It is found that splitter vanes significantly enhanced the aerodynamic performance of splitter vanes. [5] Forward curved centrifugal fan is analyzed. Increase in efficiency and pressure co-efficient has been observed by simulating various geometric configurations. [6] Man-Woong Heo, Jin-Hyuk Kim and Kwang-Yong Kim performed optimization of a centrifugal fan with implementation of splitter vanes using multi-objective optimization. Significant increase in static pressure and efficiency has been observed with implementation of splitter vanes. 3.8% increase in overall efficiency has been observed. [7] Ruizi Zhang, Kaibin Wang, Yunlong Li and Jingyin Li optimized the aerodynamic performance of a squirrel-caged fan. It was showed by both experimentally and numerically that totally pressure has been significantly improved in optimal design. [8] Davood Khoeini and Mohammad Rez Tavakoli enhanced the performance efficiency of a pump with implementation of a splitter vanes. They showed both experimentally numerically that 7.5% efficiency increase has been observed in a pump.

In the light of literature review, it’s highlighted that considerable research has been conducted on implementation of splitter vanes. For all kinds of applications varying from fans, blowers and pumps. However, applications of fans are limited to

HVAC systems only. As HVAC systems incorporates highly effective volutes; it's more like for efficiency to increase. However, this research focuses on a different type of impeller blade. Its application is based on vehicles, for engine cooling systems. Further added, blade is diagonally cut into half. Impeller blade is graphically illustrated in following figures. No simulation has been done on this type of impeller blade with such a unique area of application.

1.3 Thesis Objectives

The objective of this research is to analyze the aerodynamic performance in a backward curved centrifugal fan. Optimize the fan performance using commercial software, i-e ANSYS CFX. Details of objectives are mentioned as under:

a. To model reference geometry, validated its design using experimental work.
b. To analyze the aerodynamic performance and flow patterns of air through centrifugal fan.
c. To vary its flow and determine overall performance curve of a reference geometry.
d. To study the effect of following parameters at specific design points:
    i. Number of main blades
    ii. Length of splitter vanes
    iii. Angle between main blade and splitter vanes
e. To obtain a geometry with improved performance efficiency.

1.4 Methodology

Literature Review

Literature review is done in order to identify the research gap and margin of improvement. A total of twelve scientific papers were shortlisted. All shortlisted scientific papers have similar work to our research objective.

Experiment

Experiment is conducted in order to observe the direct values on vehicle. These values are required for validation purpose of reference geometry. These values are also required in order to establish the fact that our numerical simulation has very negligible tolerances than that of experimental values.

Fan model and Simulation

Model of impeller blade was done in Blade-Gen software. CFX was used in order to solve the equations. CFX-Post is used to post processing of results.

Reference geometry is modelled and validated w.r.t experimental results in phase one of this research. Then based on literature review, now models are prepared for analyses. Three geometric parameters are used for overall optimization of the fan.

Results and conclusions

Based on the numerical simulation, critical argument is done in order to assess the best performing model and analyze the effects on performance after the aerodynamic optimization.

1.5 Outline of Thesis

Current work comprises of six number of chapters in total. The first chapters give the introduction and the overview of this research. It also discusses the past work done in this field, research gap and future prospects of this research. Chapter two describes in detail different aspects of a centrifugal fan. It explains difference between designs of a centrifugal fans along with its working principle and nomenclature. It also argues on the parameters effecting performance and efficiency of a centrifugal fan. Chapter three is highlighting the governing equations used in numerical tools. Equations used for calculating design parameters are also discussed. Chapter four is based on the simulation setup. In chapter five, details of results and conclusion is discussed.

CHAPTER TWO

2 Theory

Fan comprises of various components. In order to fully understand the aerodynamics of a fan, it is necessary to understand the design parameters and type of applications. Understanding of these principles will help to achieve desired output from a machine. A standard isometric drawing of a centrifugal fan in as under. [3]

2.1 Design Objective of Fans

Objective of a turbo-machine is to increase energy of a gas or fluid. It can also be used for extracting energy from fluids. A machine which adds the energy is categorized as a pump. and a machine which extracts the energy from gases/ fluids is called a turbine. If a pump is designed for adding energy to gases (air) it will be called as a blower or compressor. These classifications are based on principle operation of each machine. [9] [2]Categorizations of machines is based on various other factors as well. For example, housing of an impeller. Which can be open, closed or semi-open. ASME has also categorized fans based on their performance parameters. Differentiation in each category is as following:

Sr	Type	Pressure Ratio	Pressure Rise mm Wg
a	Fan	=< 1.11	1136
b	Blower	1.11 to 1.20	1136 to 2066
c	Compressor	>= 1.20	--

Table 2.1 Comparison of Blower, Fan and Compressor

2.2 Classifications of fans

All fans are broadly classified into two types of fans based on the direction of fluid flow. These are classified as axial fans and radial fans. These fans are further classified based on the geometry of blades, housing and operational ranges of each type of fan. Classificatioon is elaborated as under:

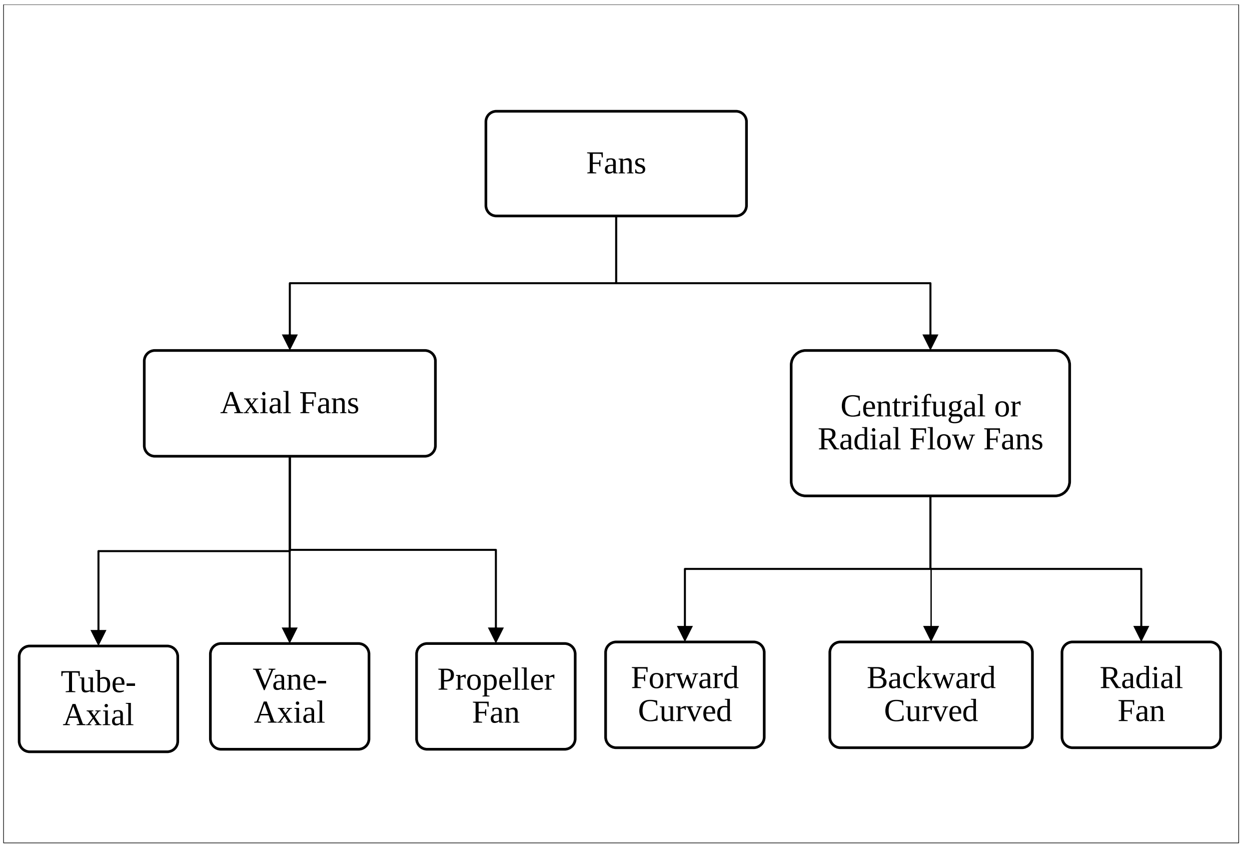


Figure 2.1: *Distribution of Fans, Blowers and Compressors*

2.3 Centrifugal Fans

Blades of a fan are of different types. Different types of blades are as following:

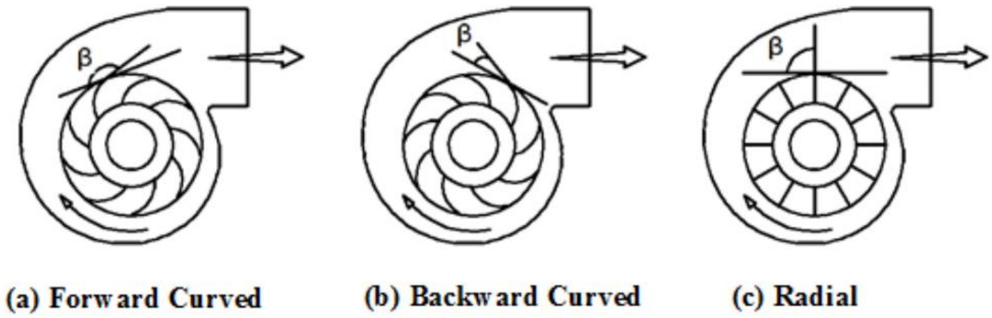


Figure 2.2: *Illustration of forward curved, backward curved and Radial fans*

Forward Curved

Trailing edges in these types of fan blades are bent forwards in the direction of rotation. On average, a forward curved fan may comprise of 24 to 64 number of blades. At the TE of blade, fluid flow velocity is maximum. In other words, wheel peripheral velocity is less than air tip velocity in case of forward curved fans. [13]

Backward curved

Trailing edges in these types of fan blades are bent backwards in the direction of rotation. A backward curved impeller is usually composed of 6 to 18 number of blades. Backward curved fans are well known for their efficiency. As these fans reach at max power and then its power requirement falls under usable air flow range. [13]

Radial blades

Leading edge to Trailing edges is a straight line in radial direction. These fans are also composed of 6 to 16 number of blades. These types of fans are known as industrial rugged fans and are used for various applications.

This research focuses on backward curved centrifugal fan. [13]

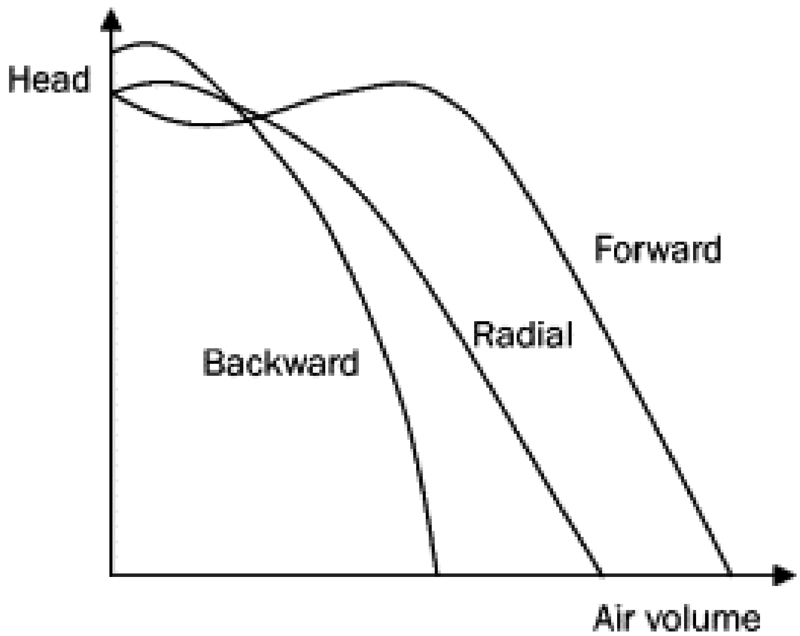


***Figure 2.3**: Performance Curve of Fans [13]*

2.4 Significance of a Cooling System in Vehicles

Cooling system in a vehicle is of great significance. Cooling system controls and maintains engine temperatures. A combustion engine operates at higher temperature in order to develop more power. Due to elevated temperatures in an engine, vital components undergo wear and tear. This wear and tear are minimized with the use of lubrication oil. [13] During operation of a combustion engine, temperature of lubrication oil will also rise. Hence, for a continuous operation, systems are developed in order to minimize the engine oil temperatures. Efficient cooling system will not only reduce the fault trends, rather it would enhance the overall life span of an engine. The cooling system is of the liquid, high closed type with the forced circulation of the coolant and ejective cooling of the radiators.

2.5 Components of a Cooling System

Cooling system comprises of multiple assemblies. i-e, forced fan, water pump, oil pump, water radiator and oil radiator etc. Each assembly itself has margin of improvement for performance and efficiency enhancement. There are multiple variables or parameters associated with each assembly, which can be optimized for overall performance enhancement of a system. However, in this study forced fan is under consideration. List of components associated with cooling system is following:

a. Radiators.
b. Oil Coolers
c. Surge tank.
d. Water pump.
e. Oil pump. [13] [10]

2.6 Operation of a Cooling System

During normal operation of a vehicle, the forced fan is driven taking power through a shaft linked with gear box. As large volume of air is passed from the forced fan; vacuum under the radiators is created. Under the effect of vacuum the outer air passes through the radiators cooling them, and then discharged at the outlet. Simultaneously, the coolant is being circulated by means of engine water pump. Coolant comes from water pump to engine cavities and cools the cylinders. All coolant from the tank is directed along a conduit to the engine water pump. Coolant temperature is controlled by an electric thermometer, its indicator being at driver's panel. [10]

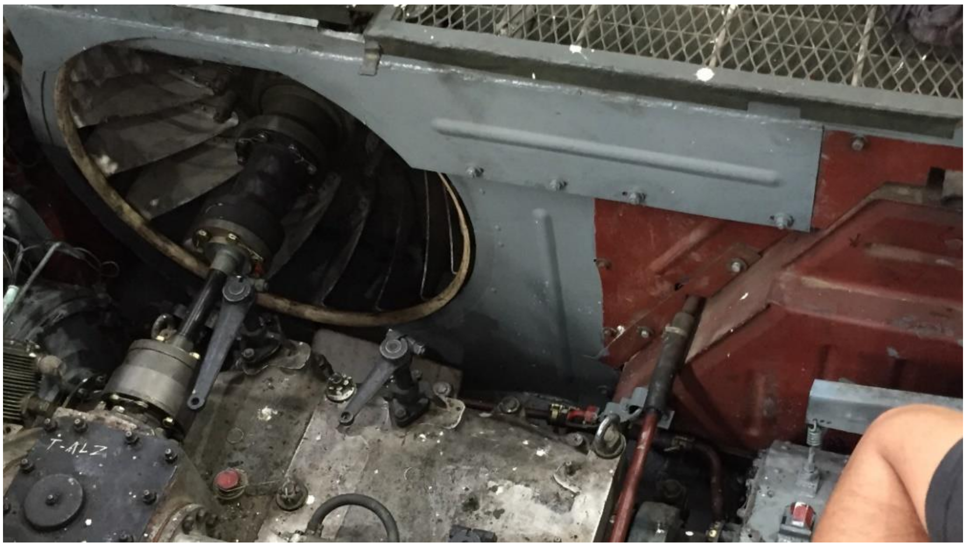


Figure 2.4: *Physical System Installed on Tank*

2.7 Backward Curved Centrifugal Fan

Centrifugal fan is an assembly in which a blade with spiral shaped tongue is fitted in a volute cavity. The working fluid is entered in the center of the blade through the direction of axis of rotation, it exits the casing in the direction of periphery at the trailing edge of blade. The amount the work done required by a centrifugal work to perform efficiently is directly related to the pressure difference. [13] [10]

Figure 2.5: *Reference Geometry (Impeller)*

2.8 Working Principle

The working principle of a centrifugal fan is pretty much depicted from its name. Which is “centrifugal’. The word itself means a circular motion with, with acceleration directed towards the center of motion [13]. A typical example of a centrifugal motion can be compared with that of a stoned attached with a string, and swung. A centrifugal fan operates in a similar fashion. It moves the air outwards in a circular motion and throw the air towards its trailing edge. As the fluid is pushed towards the outlet duct, more fluid is entered through the inlet duct and the procedure is continued. [13]

It is the function of a volute shape of a casing that acts along with the blade in order to add the pressure head to the fluid.

2.9 Vector diagram

As already discussed, the types of fans. Each type of fan has particular application and advantage of its design. The velocity of air at trailing edge varied with the type of blade. Vector diagram is shown as under.

The horizontal component V1 demonstrates the tangential direction of the flow whereas vertical component V2 shows the direction of radial component of the flow. It can be observed that the resultant component of in the forward curved blades is higher than others. Whereas in case of backward curved fans, resultant is lowest. In case of radial fans, resultant is being in between than those of forward and backward curved fans.

2.10 Nomenclature

A centrifugal fan comprises of following assemblies and items:

a. Rotor
b. Hub
c. Shroud
d. Inlet duct
e. Outlet duct
f. Volute casing
g. Volute tongue

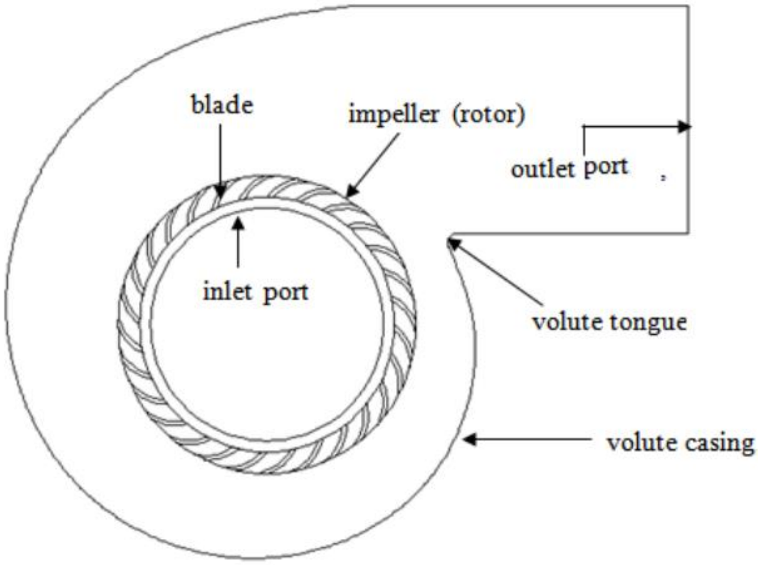


Figure 2.6: *Nomenclature of a Centrifugal Fan*

2.11 Volute casing

Volute casing covers the rotor. Rotor is mounted on a volute casing. Volute is a stationary assembly that guides the air to enter and exit through the rotating blade. Volute functions by increasing the pressure head by reducing the KE of fluid flowing through the rotating blade. A fan with no volute may generate the streamlines of vortex flow [13]. Hence, a fan with a volutes casing will have similar spiral streamlines inside. Hence, it's recommended to design the volutes curves in similar contours in order to have constant pressure at outlet. This also minimizes the radial thrust in driving shaft and bearings. [8]

2.12 Blades

Blade is a component which adds the pressure head to the working fluid by reducing its KE. This is done by mechanical energy provided by shaft work. The pressure head causes the working fluid to exit the volute casing.

2.13 Inlet and Outlet Duct

Inlet ducts acts to streamline and channel the air flow into the volute casing. Pressure head is added with the help of mechanical energy into the flow. Outlet duct allow to exit the flow. These three assemblies act simultaneously to increase pressure head at a uniform and constant contour.

2.14 Design Parameters

The primary objective of any type of fan is regarded as to pass mass flow at constant pressure. There are different types of pressures which are produced during any type of fan operation.

Static Pressure:

Static pressure is defined as the difference of total pressure and outlet pressure. Static pressure can either be positive of be negative. [6]

Dynamic Pressure:

Dynamic pressure is defined as the average of flow speed at the discharge location of outlet duct. Dynamic pressures are positive. Dynamic pressure is caused by the motion of flowing medium. [6]

Total Pressure:

Total pressure is defined as the algebraic sum to dynamic pressure and the static pressure. Total pressure will be equivalent to static pressure when the rotor blade is in static position. [6]

2.15 Non-Dimensional Parameters

Few design parameters are of great interest for modifications and improvements in design. Such design parameters are converted into dimensionless numbers. These dimensionless numbers enable engineers to have an insight by comparison of characteristics. Commonly used non-dimensional parameters for a fan performance analysis are as under:

a. Flow Co-efficient

$$\emptyset = \frac{Volume\ Flow}{\left(\pi\, D^2/_4\right) u}$$

b. Total Pressure Co-efficient

$$\psi = \frac{p_{total}}{\left(1/_2\right) \rho\, u^2}$$

Given that u shows the impellers peripheral velocity ($^{\omega D}/_{2}$). Where D is the inflow diameters and omega show the rotational speed of impeller. [6] [5]

2.16 Losses in Fan

In real life, there are several losses when a fan operates. These losses account for pressure/ volume curves under the corresponding theoretical results. Few of the common losses that occur in fan are as under:

Frictional Losses:

Frictional losses are causes by the interface between body surface-to-fluid and fluid-to-fluid. It is also termed as surface friction. Which is caused by the rough surface in volute inlet, volute outlet and blades passage. Frictional forces also exist between the different layers of flowing fluid medium. This type of friction varies between different types of fluid flows. [5] [6]

Eddy Losses:

The pressure on the higher side of the periodic is greater than that on the lower side of the periodic. Consequently, the air flow velocity at trailing edge is slightly greater than that in the higher side of the periodic. This phenomenon accounts for the asymmetric behavior. This causes the velocity of fluid medium to be non-homogeneous in between two adjacent blades. The power distribution is non-homogenous along the periphery of blade tongue. [5] [6]

Shock Losses:

As soon as the flowing medium fluid enters the rotating passage of the blade, fluid experiences sudden turn at inlet. This behavior causes the shock loss. Inlet blade angles can be optimized in order to reduce the shock losses. [5] [6]

2.17 Performance Curves

A fan curve represents the overall performance for a particular fan at specific design points. It is a representation of number of inter-related parameters. A typical fan curve is plotted based on the parameters like volume flow rate, static pressure, rotating speed of the impeller blade etc. [9]

CHAPTER THREE

3 Mathematical Model

A fan has variety of real-life applications, as already discussed in chapter-one of this research. Fans are utilized for a variety of applications like HVAC syst, air supplies in industries, chemical industries, consumer electronics and process industries. Its significance cannot be denied. The governing equations that are being solved by CFX solver are discussed in this chapter. All these equations are sourced from ANSYS CFX solver manual.

3.1 Assumptions in Model

In order to effectively solve the equations in suitable time period, various assumptions are made, these assumptions efficiently simplify the governing equations. Governing equations make it possible to capture the physical phenomenon.

Incompressible Flow

Fluid flow is considered as incompressible. It is because of the fact that fluid velocity is typically low for a centrifugal fan.

Isothermal

All kind of heat transfers into or out of the system are neglected. There is no heat added or removed out of the system,

Gravitational Effects

As the gravitational effect is not considered in this research, all the body forces associated with the fluid flow are neglected. Medium is also considered as Newtonian fluid.

Fluid Properties

Change in temperature is negligible, hence any sort of heat added or removed from the system is neglected.

3.2 Governing Equations

ANSYS CFX numerical package provides the comprehensive tool for analyzing complex engineering problems. CFX package is capable to solving both compressible and incompressible problems. Both type of simulations can be performed, that is for laminar or turbulent fluid flow. Analysis like steady state and transient can be performed.

The governing equations are formulated by mathematical modelling physical fluid flows. These equations are formulated in inertial frame of reference. These equations can also be converted in rotational frame of reference based on requirement.

ANSYS CFX solves the conservation equations for Mass and Momentum. As in this research, our objective is to perform steady state simulations and temperature effects are neglected. Hence, energy equations are not used in the study.

3.3 Mass Conservation Equations

The continuity equation is described as follows:

$$\frac{\partial \rho}{\partial t} + \Delta . (\rho \, \vec{v}) = s_m$$

The general form of mass conservation is described as above. This equation is valid for almost every type of fluid flow. The term defined as (source) s_m is the mass added to the continuous fluid. As we have already discussed the scope of this research is focused on steady state simulations, hence transient term can be eliminated from the equation. The reduced form of equation is as following:

$$\Delta . (\rho \, \vec{v}) = s_m$$

3.4 Momentum Conservation Equations

The conservation of momentum equation in inertial FOR is defined as following:

$$\frac{\partial(\rho\,\vec{v})}{\partial t} + \nabla.(\rho\,\vec{v}\,\vec{v}) = -\nabla \mathrm{p} + \nabla.(\vec{\tau}) + \rho\,\vec{g} + \vec{F}$$

Where each variable is

p	Static Pressure
$\vec{\tau}$	Stress Tensor
$\rho\,\vec{g} + \vec{F}$	Body Forces

When the above-mentioned equation is used for a centrifugal fan, it is reduced as following by eliminating terms as:

$$\nabla.(\rho\,\vec{v}\,\vec{v}) = -\nabla \mathrm{p} + \nabla.(\vec{\tau})$$

The stress tensor $\vec{\tau}$ is defined as:

$$\vec{\tau} = \mu\left|(\nabla\vec{v} + \nabla\vec{v}^{T}) - \frac{2}{3}\nabla.\vec{v}\,I\right|$$

Each variable is as under

μ	Molecular viscosity
I	Unit Tensor
$\frac{2}{3}\nabla.\vec{v}$	Volumetric Dilation

3.5 k-ε Models

There are three k-ε models for solving simulations. Their names are as under:

- Standard
- RNG
- Realizable k-ε Model

All three have same forms of transport equations. The difference in these is based on following aspect of the equations.

- Approach for calculating turbulent viscosity
- Turbulent Prandtl numbers
- Generation and destruction terms in the ε equations

However, we will discuss Standard k-ε Model here. As all of our results are obtained by solving standard k-ε Model.

Standard k-ε Model

The most simplified model used in most of the numerical tools available is Standard k- ε model. It is based on two turbulent model equations. Solution of two separate transport equations is allowed. The turbulent velocity and length scales can be determined independently. This model is semi-empirical model that is formulated on transport equations. This model is also formulated for the turbulence kinetic energy-k and dissipation rate-e. The transport eq for k is derived from same eq, whereas for e its obtained using physical parameters.

In the derivation of the k-e model, following assumptions are made:

- The flow is fully turbulent.
- Molecular viscosity is negligible for the flow.

CHAPTER FOUR

4 Validation Work

4.1 Experiment

In order to predict he fan performance; various analytical and numerical tools are available. But even advanced techniques required to be validated with an aid of empirical results. The experiments have always proven to be the most accurate way of establishing facts related to validity and performance, Number of performance values were observed experimentally in order to establish the fact that our 3d modelled geometry is close to physical geometry.

4.2 Objectives of Experiment

The objectives of experimental observations are to validate our numerical results. Determine the particular design points at with all the modified geometries can be evaluated.

4.3 Procedure

The machine on which experiment is to be performed, is actual reference geometry. i-e Back-Curved Centrifugal Fan. Fan is driven by a mechanical shaft linked with gear box. The gear box is driven by engine. Speed of fan is directly proportional to engines RPM. The total number of blades on centrifugal fan is 18. The inlet blade angles are measure to be 115 degrees whereas outlet blade angle is measured as 150 degrees.

Sr	Description	Value	Unit
a	Blade Number	18	No.
b	External Diameter	780	mm
c	Internal Diameter	420	mm
d	Inlet Blade Angle	115	Degree
e	Outlet Blade Angle	150	Degree
f	Blade Shape	Circular arc	-
g	Blade Thickness	3	mm

Sr	Description	Value	Unit
h	Blade Height	190	mm

***Table 4.1:** Specifications of Impeller Blade*

The engine was operated at standard atmospheric pressure and temperature. The fluid medium is fresh air. The average air temperature while performing the experiment was 27-degree Celsius. The air standard density is taken as 1.225 kg/s.

Digital anemometer is used to calculate the flow velocity at the outlet. The flow discharge area was sub-divided into four sections, as shown below:

***Figure 4.1:** Points of observation of air flow velocity*

At each point several values were observed and noted down against angular velocities. The angular velocity of a fan is calculated using a tachometer. A reflective duct tape was sticked to the blade. The infrared laser from tachometer is used to calculate the rotational speed. The three rotational speeds of fan were selected as maximum, minimum and at in between point for analysis.

4.4 Observations of Experiment

All data points measured were averaged. Hence the average flow speed of air was observed against each angular velocity. The details of data observed are as under:

Experimental Results						
Fan RPM	Point A	Point B	Point C	Point D	Velocity [kph]	Velocity [m/s]
756	12.44	27.12	29.14	23.61	23.08	6.41
1030	25.95	56.52	64.93	42.53	47.48	13.19
1398	53.66	70.03	90.03	74.00	71.93	19.98

***Table 4.2**: Experimental Observations of Air Flow Velocities*

The average of flow velocity is 19.98 m/s at 1398 rpm of the fan.

4.5 Calculations

Now the average mass flow rate can be calculate using following equation.

$$\dot{m} = \rho \; A \; v$$

The dimensions and area of discharge surface is calculated as following.

Sr	**Properties**	**Value**	**Unit**
a	Density of air	1.225	m^3 / kg
b	Length	1200	mm
c	Width	170	mm
d	Area	0.204	m^2
e	Flow Velocity	19.98	m/s
f	Mass Flow Rate	4.993	kg/sec

Table 4.3: *Values used for calculating mass flow rate*

The rounded off value for the mass flow rate at the flow discharge location is 5 kg/s. Similarly, the angular velocity is rounded off as 1400rpm.

CHAPTER FIVE

5 Numerical Simulation of a Centrifugal Fan

5.1 Fan Model and Simulation

ANSYS CFX code is used to solve the problem setup. ANSYS CFX utilizes finite-volume method in order to solve the numerical model. ANSYS CFX is capable of solving various complex engineering problems related to incompressible or compressible flows. Even if the flows are in inviscid or viscous, laminar or turbulent, ANSYS CFX can incorporate all the properties in physical model.

ANSYS CFX solves the governing equations of problem by either pressure-based solver or density-based solver. The pressure-based solver is modeled for low-speed incompressible flows. Whereas density-based approach is formulated for high-speed compressible flows.

As far as this research is concerned, centrifugal fans are based on low speeds of flow. Also, the flow is assumed as incompressible. Hence, the most suitable approach for solving governing equations for this research is pressure-based solver.

This chapter presents the setup of CFX simulation for both base geometry and the reference geometry. It discusses detailed problem-solving algorithm. Fan blade geometry is modelled using Blagden and mesh is produced using T-grid. Pre-Processing and Post-Processing is also done by ANSYS CFX.

5.2 Modifications of the Reference Geometry

Efficiency of the fan is a measure of how well a fan is moving volume of air at a certain pressure. A turbulent or non-uniform fluid flow is responsible for a less efficient fan. Therefore, a fan design must include a proper streamlined geometry for inlet and outlet ducts. As a fan equipped with streamlined accessories will allow more fluid to pass and will significantly enhances the efficiency.

Generally, efficiency of a fan has strong correlation with performance parameters like pressure, volume flow rate and power absorbed. The most critical geometric parameters that effect the fan performance are listed as under:

- Make and Type of Geometry
- The Size of the fan
- Number of main blades
- Tongue length of main blades
- Angle of main blades

5.3 CFD Simulation Setup Algorithm

The overall problem-solving flow is elaborated as under:

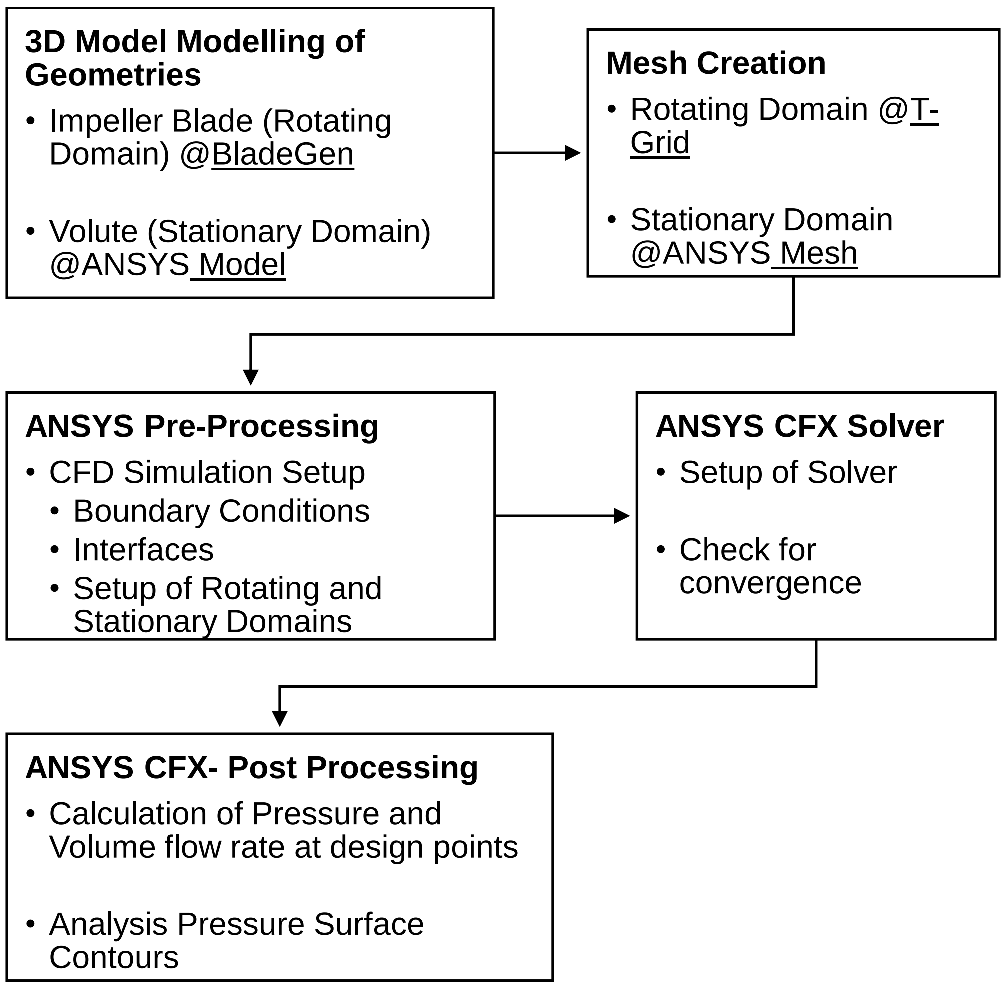


Table 5.1: *Flow Diagram for CFD Simulation*

5.4 Construction of 3D Model

Blade-Gen is a package provided by ANSYS in order to model all sorts of blades pertaining to turbines, compressors, blowers and fan. It is very powerful tool, that enables the user to model rotating domains by entering geometric parameters. Complex geometries can be modified by simply interactive graphs and curves. Blade Gen software produces a curve file for each hub, shroud and blade. These files are readable by T-Grid software, for mesh creation.

Stationary domain of centrifugal fan, i-e volute, was modelled in ANSYS Model. It is a powerful tool for creating refined and clean geometries for analyses.

5.5 Mesh Creation

All equipment's related to turbo machineries are divided into two categories. i-e Rotating domains and stationary domains. Rotating domain is based on impeller blade whereas the stationary domain is based on the volute cavity/ scroll.

Stationary Domain:

With triangular elements, mesh is created for volute cavity independently in ANSYS Mesh. This domain collects the information related to fluid leaving the rotating domain and being discharged at outlet. It also converts the kinetic energy of the fluid into the pressure head. As our primary focus is based on the geometric parameters pertaining to splitter vanes. The volute cavity is not a focused in this research. Hence, coarse mesh is applied in this domain.

Rotating Domain:

Mesh for rotating domain, i-e impeller blade I generated in T-Grid. The quality of mesh is Hexa-Mesh. This domain is of great significance. As major energy transfer takes place in this domain through fluid medium. In order to capture maximum details of fluid flow through impeller passage, this region must have high quality and resolution of mesh. Hence, higher grid number is placed in impeller passage.

5.6 Boundary Conditions

Boundary conditions specifies the physical conditions at inlet, outlet and interfaces. Boundary conditions defines the operational characteristics of the model. The boundary specifications implemented in this research is appended below:

Sr	Description	Boundary Type
a	Inlet	Pressure Inlet
b	Outlet	Mass Flow Rate
c	Impeller Blades	Wall- No Slip
d	Volute	Wall- No Slip
e	Rotary to Stationary Domain	Stage Mixing

Table 5.2: *Boundary Conditions used for Simulation*

5.7 Defining Domains

As discussed earlier, our model is based on two distinct domains. That is stationary and rotary domain. Fluid through these domains have to be defined as incompressible and steady state flow. These flow models are used to proceed with simulation for solving the governing equations.

The wall boundary conditions limit the motion of fluid. This boundary condition is applied at all surfaces of volute, bounding the fluid and at external surfaces of impeller blades.

5.8 Defining Operations

The fluid used for simulation is ideal air at STP conditions, pre-defined in ANSYS CFX. The stationary domain is fully constrained. The rotary domain in rotated around Z-Axis. The angular speed for rotating frame of reference is variable. As it is discrete design point. Angular speed is varied between 750 RPM to 1400 RPM.

5.9 Execution of Solver

Graphical monitors are defined in order to monitor control variables and residuals throughout the solution. In this study, mass and momentum residuals, pressure at discharge point and volume flow rate at discharge point are monitored. Residuals are set to 1E-5 and 1000 no. of iterations are executed for each set of boundary conditions.

5.10 Design Point and Boundary Conditions

The design point for the analysis and optimization of geometry is as under:

Sr	Description	Property	Value	Unit
a	Inlet BC	Static Pressure	1	atm
b	Outlet BC	Mass Flow Rate	5	kg/s
c	Surface Condition	No Slip	-	-
d	Thermal	Isothermal	-	-
e	Rotation	Angular Speed	1400	RPM

Table 5.3: *Values of Boundary Conditions*

5.11 Analysis and Interpretation of Results

Compare the results from experimental and numerical simulations for reference geometry in order to ensure the validity. The geometric alterations are made in a verified model in order to examine the effects of modifications.

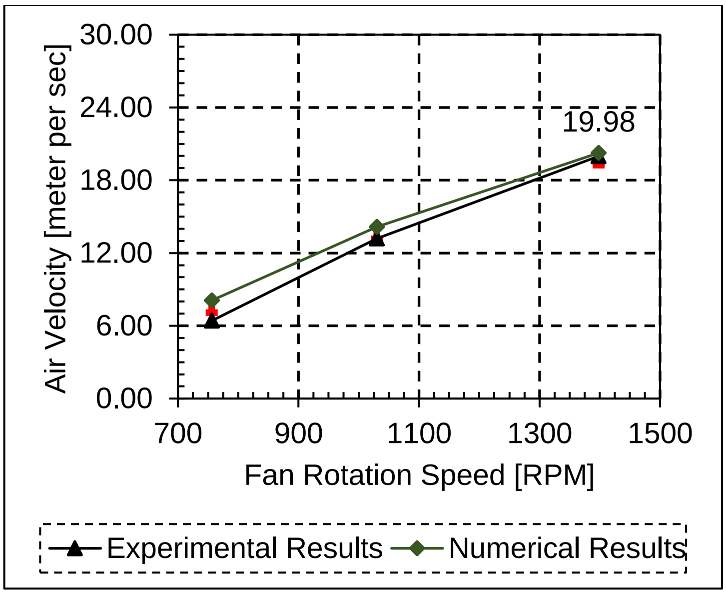


Figure 5.1: *Experimental vs Numerical values of Flow Velocities*

5.12 Post-Processing

Post processing is the final phase of CFD analysis. It involves the interpretation of numerical solution using graphical aid. Various graphical contours and vector diagrams can be plotted for each design parameter. However, it is necessary to ascertain the convergence of numerical solution. There is no standard procedure to evaluate convergence. As convergence for one type of CFD may be invalid for another type. Hence, it is recommended to monitor physical design parameters as well in addition to mass and momentum residuals. Few recommended practices in this regard are mentioned as under:

- The residuals should decrease sufficiently depending upon the complexity of problem. In this research, mass and momentum are set to 1E-5.
- Solution and design parameters/ flow variables should stabiles over the wide range of iterations.

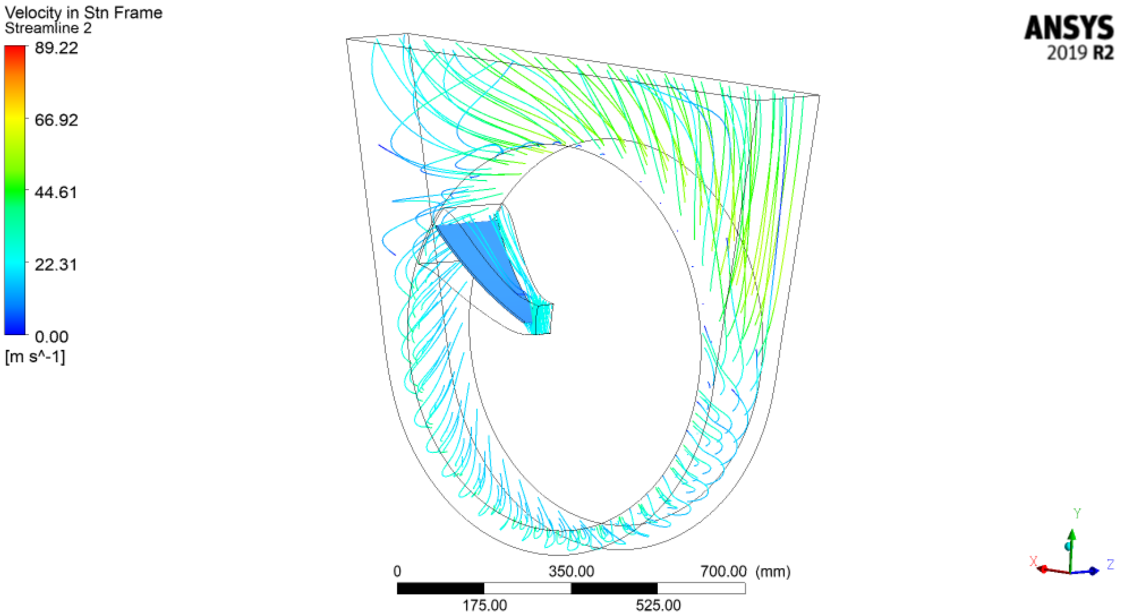


Figure 5.2: *Visual of Post Processing in ANSYS CFX*

5.13 Grid Convergence Test

Grid convergence is done in order to evaluate and verify the mesh independence of computational domain. The number of elements were increased gradually and design parameter is monitored for simulation. Number of elements were equally increased from each stationary and rotary domain. Details of grid convergence performed is appended below:

Sr	Grid Convergence Pts	No. Of Elements	Vol Flow Ratem^3/ min
a	1.00	323434	242.67
b	2.00	382940	248.28
c	3.00	490857	251.12
d	4.00	591400	253.40
e	5.00	691580	253.20

Table 5.4: *Data pertaining to Grid Convergence*

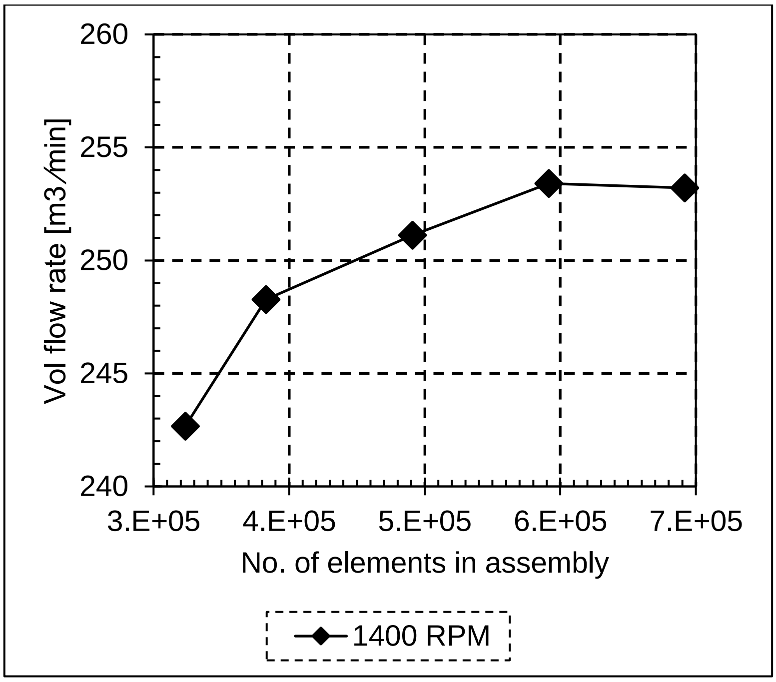


Figure 5.5 *Grid Convergence Plot*

CHAPTER SIX

6 Results and Discussions

An effort has been made in order to CFD analysis of a centrifugal fan for aerodynamic optimization. The type of centrifugal fan backward curved centrifugal fan. Experiments were also conducted in order to validate and verify the numerical results of base geometry. Base or reference geometry is systematically modified and simulated. By varying various geometric design parameters related to centrifugal fan, its performance is analyzed and best optimal design is selected.

Chapter six uses graphical aids for analyzing and discussing the flow characteristics of the fan and numerical solutions

6.1 Number of Blades

The number of blades in base geometry is 18. After varying total number of blades in base geometry, significant increase in performance is observed. Rise in static pressure and increase in efficiency by altering base geometry is appended below:

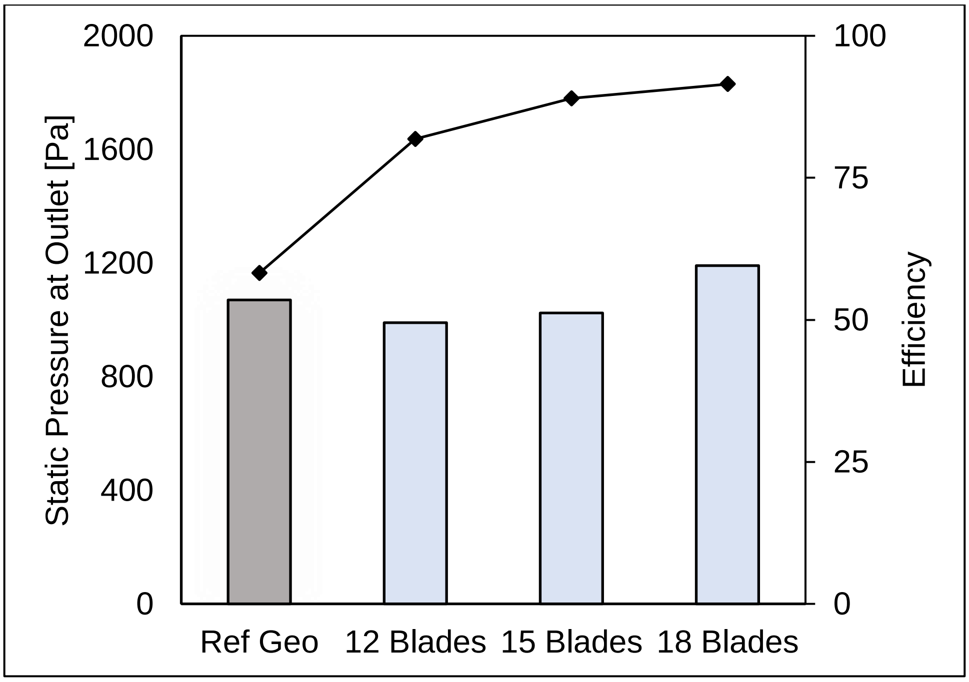


Figure 6.1: *Comparison of varying no of blades and reference geometry*

6.2 Length of Splitter Vanes

Performance of a centrifugal fan is analyzed at different lengths of splitter vanes. Design point selected is different percentages of length of splitters in terms of main blades. Comparison of performance parameters w.r.t to reference geometry is appended below:

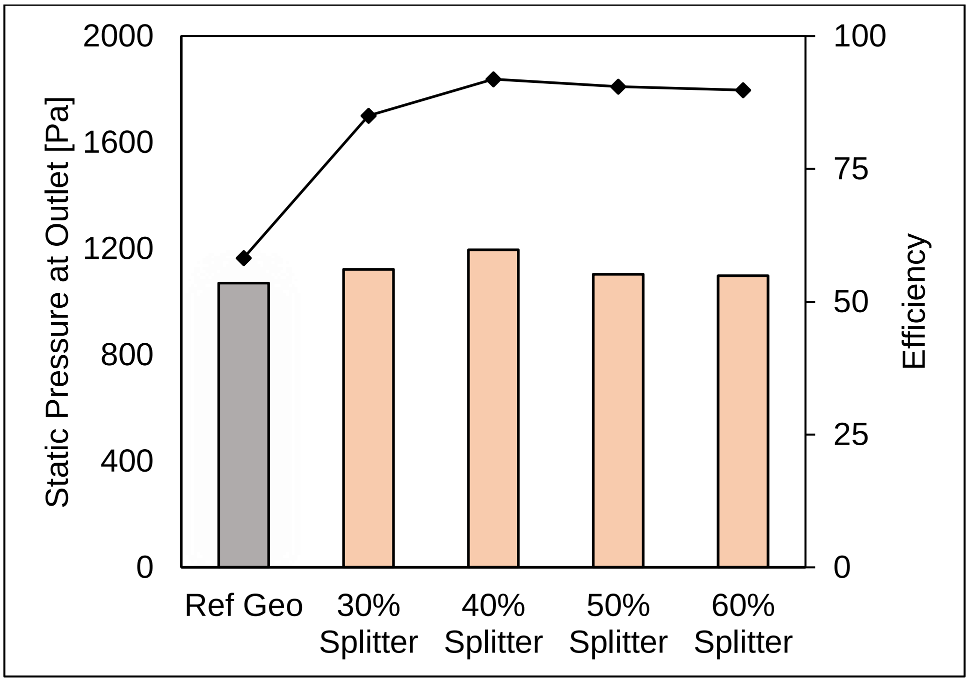


Figure 6.2: *Comparison of varying splitter vanes length and reference geometry*

6.3 Angle b/w Main Blades and Splitters

Similarly, as done in previous section, angle between main blade and splitters is varied. Results pertaining to performance parameters is recorded and plotted.

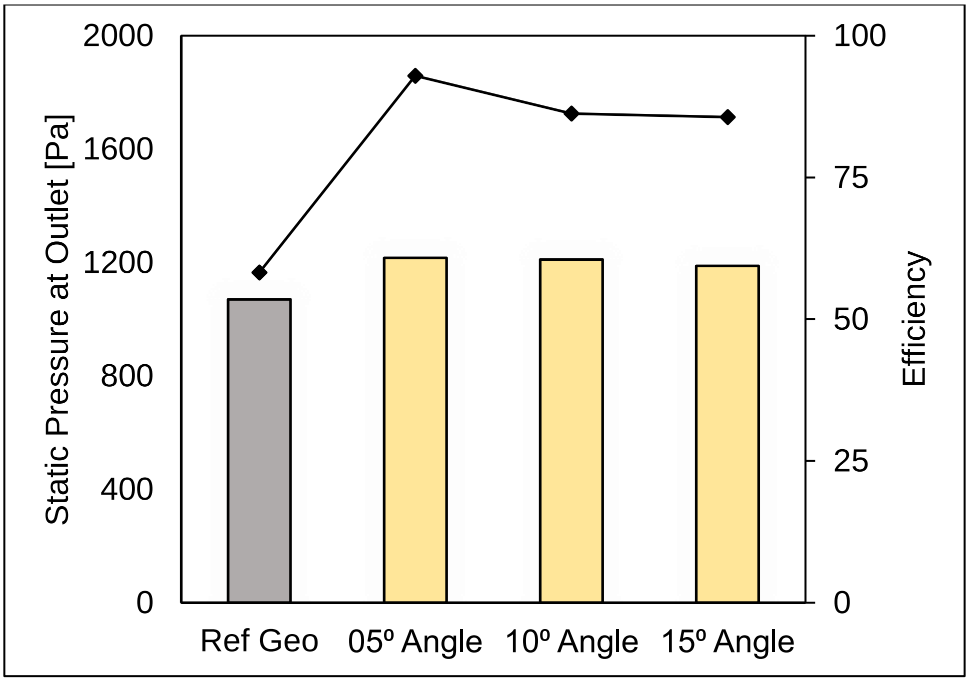


Figure 6.3: *Comparison of varying splitter vanes angles and reference geometry*

6.4 Reference Geometry vs Modified Geometry

Series of modifications have been performed for geometric parameters. All modifications have been performed in a systematic and organized manner. The best performing geometry is selected for further analyses.

The comparison of overall fan performance curve is appended in following section,

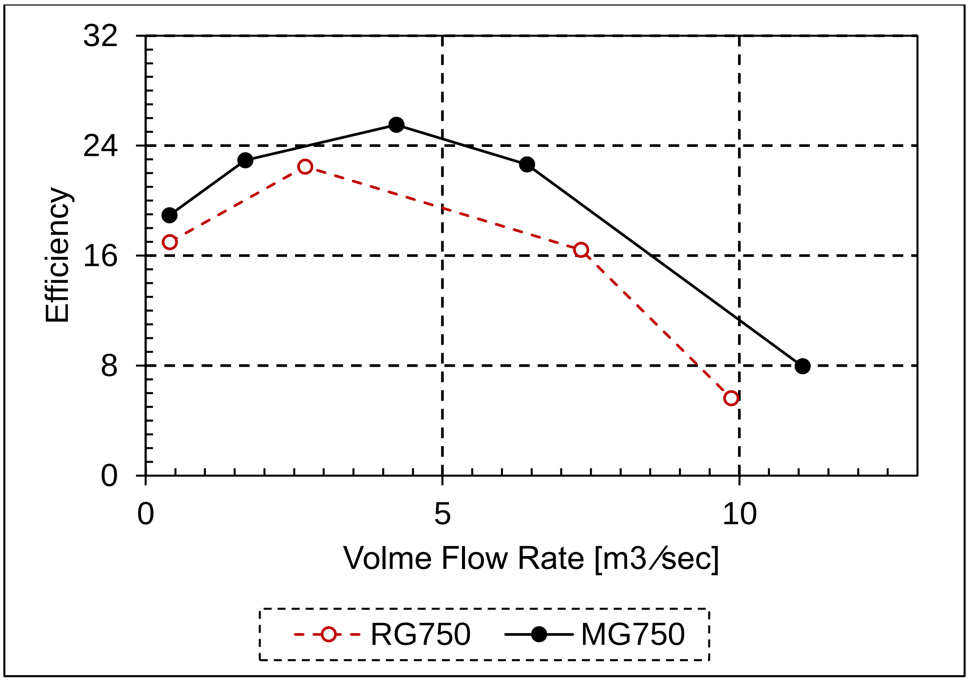


Figure 6.4: *Efficiency Plot at 750 RPM*

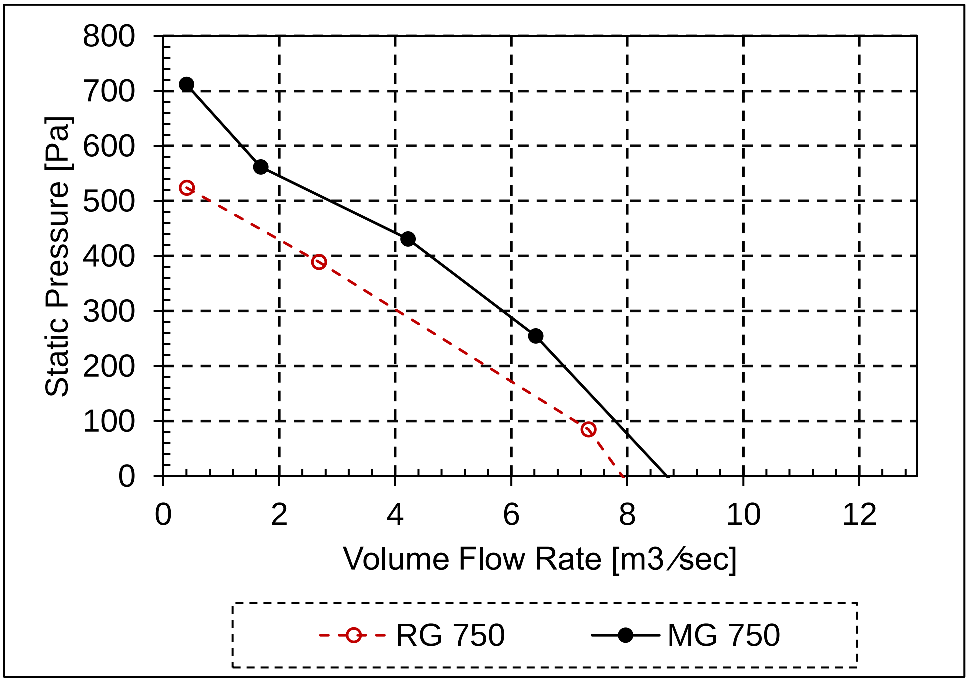


Figure 6.5: *Static Pressure Plot at 750 RPM*

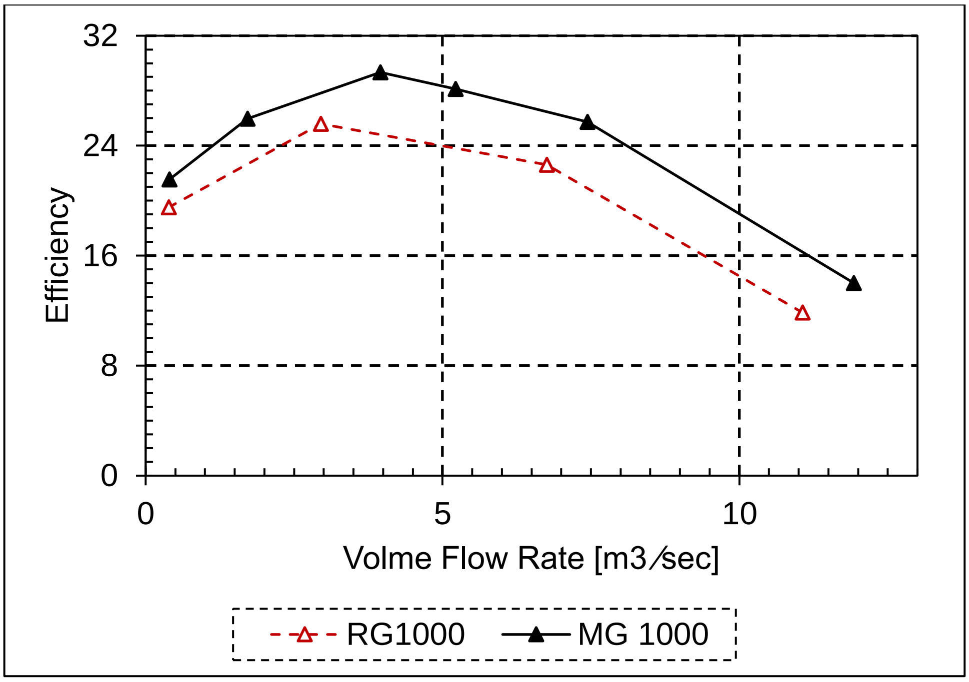


Figure 6.6: *Efficiency Plot at 1000RPM*

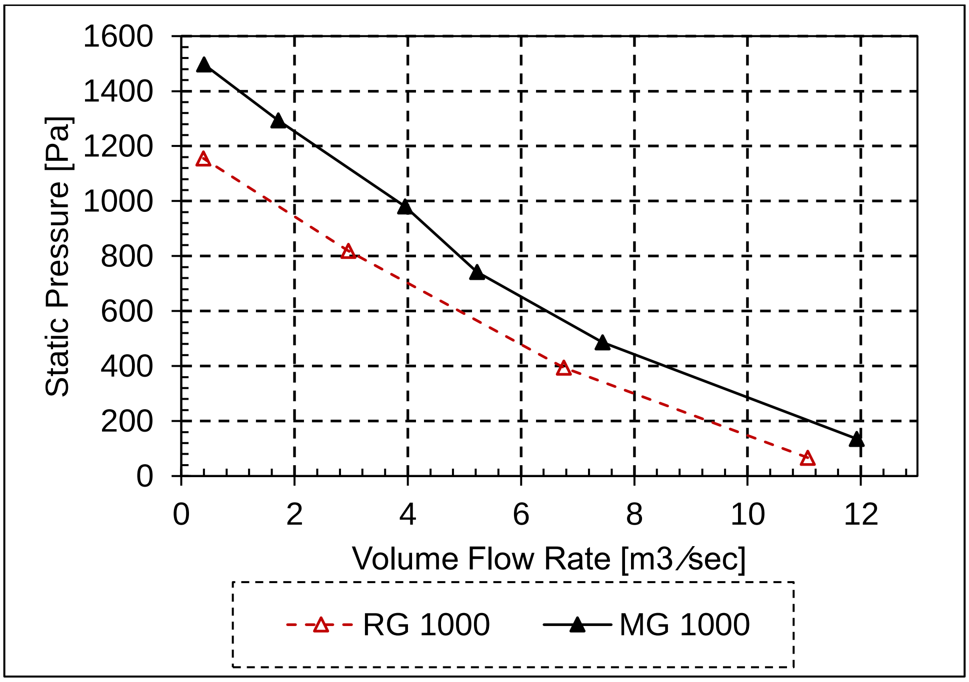


Figure 6.7*: Static Pressure Plot at 1000RPM*

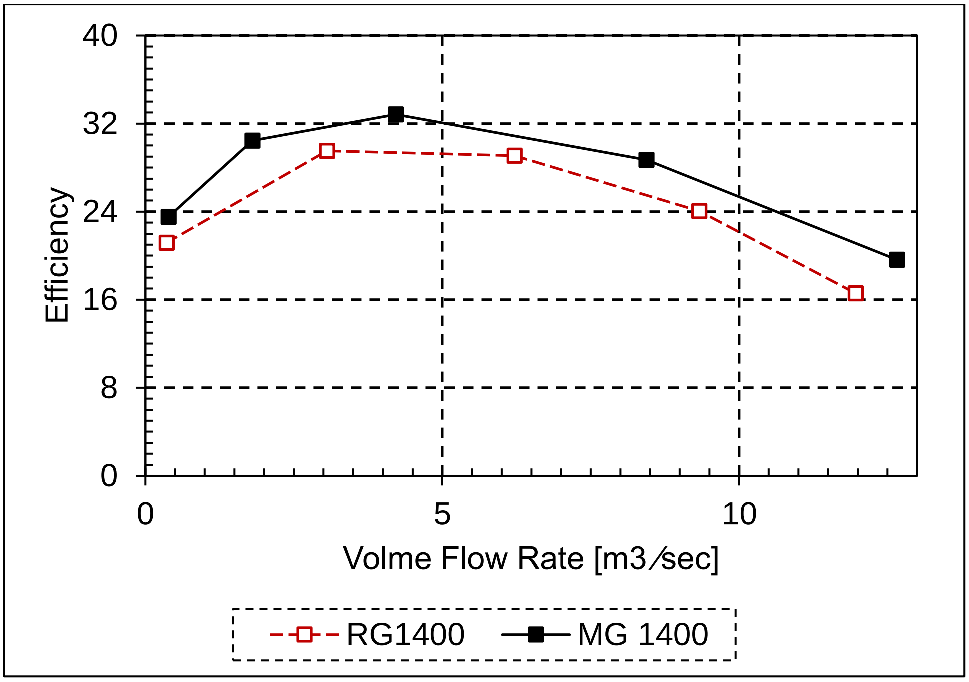


***Figure 6.8** : Efficiency Plot at 1400RPM*

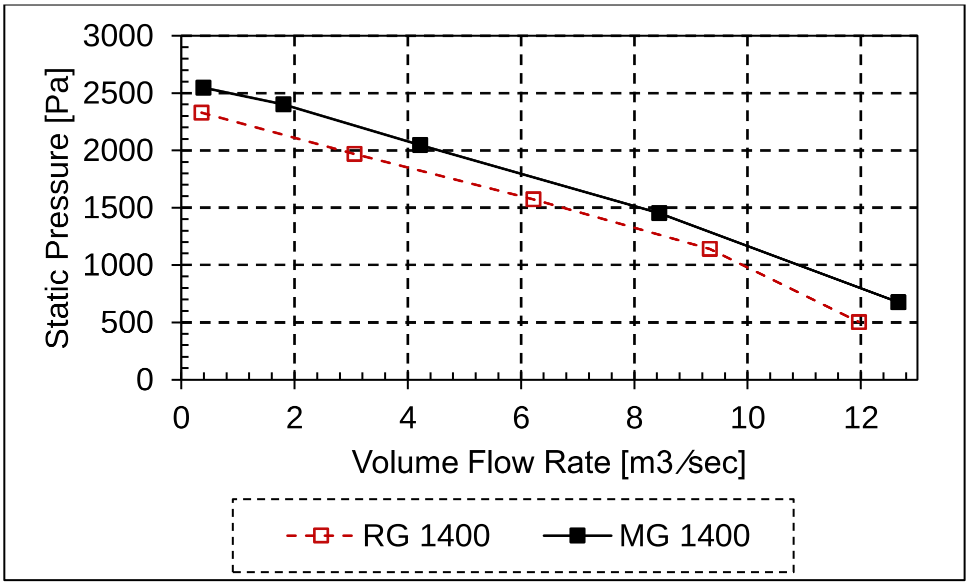
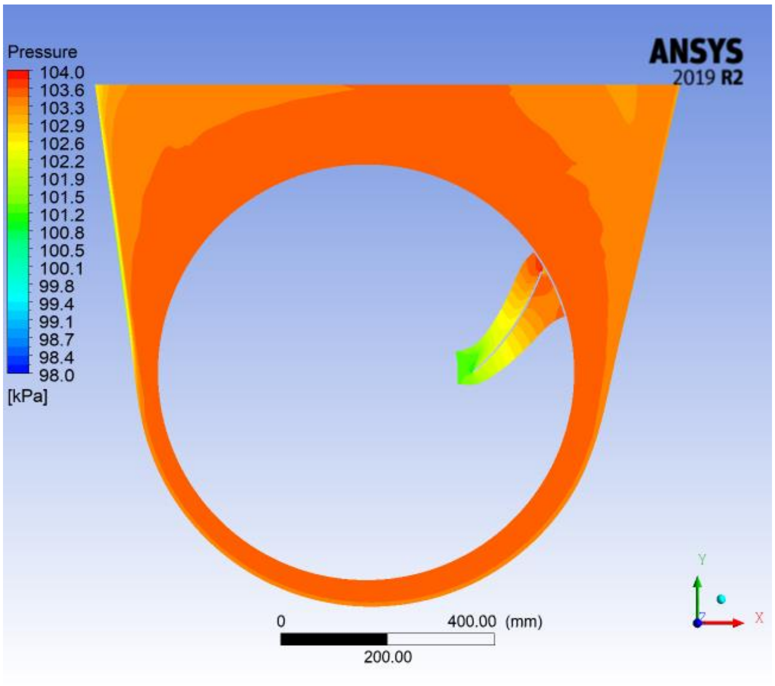
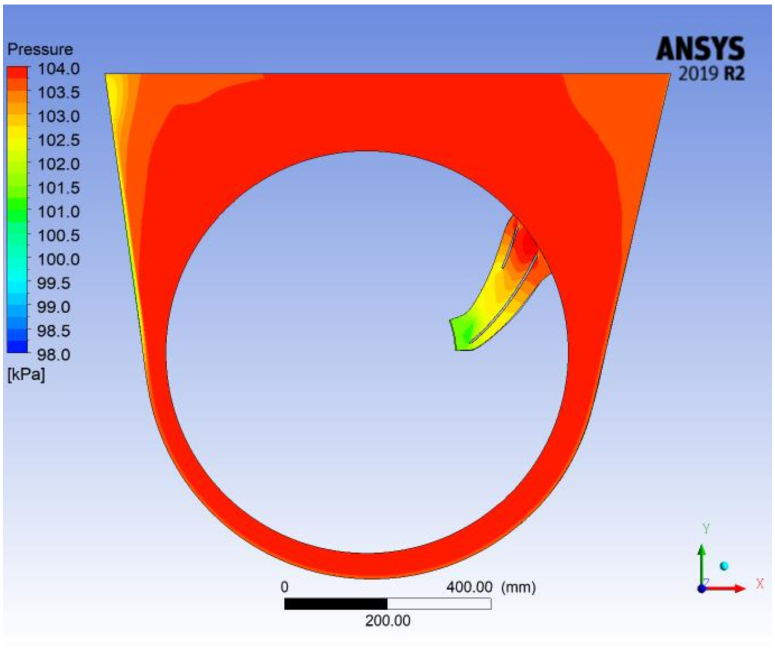
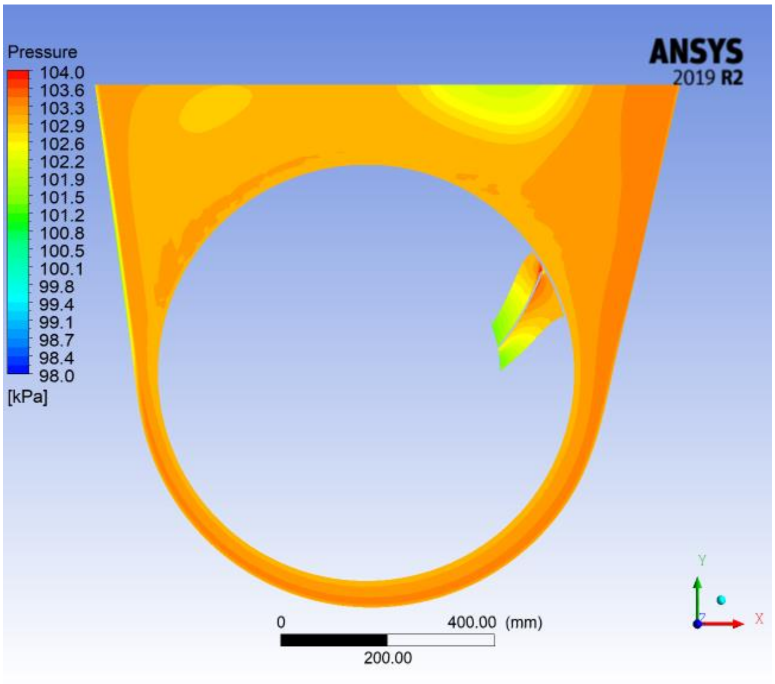
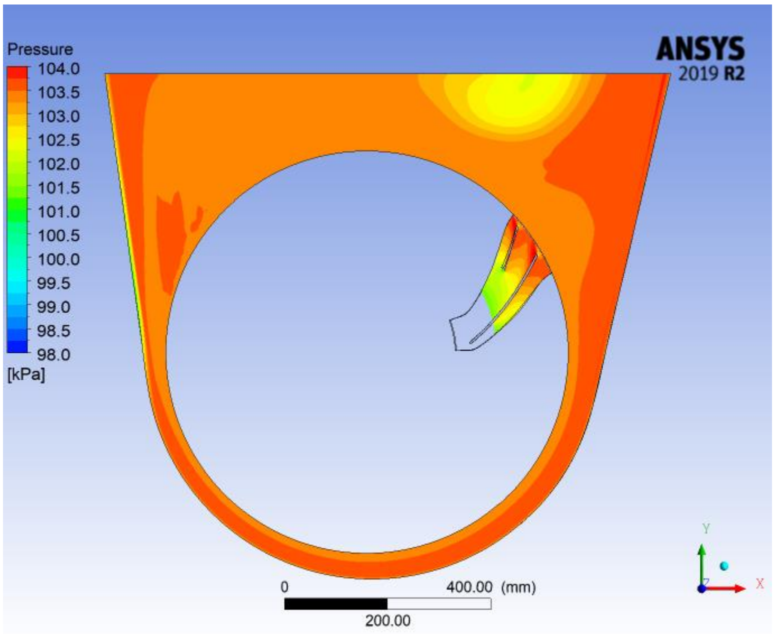


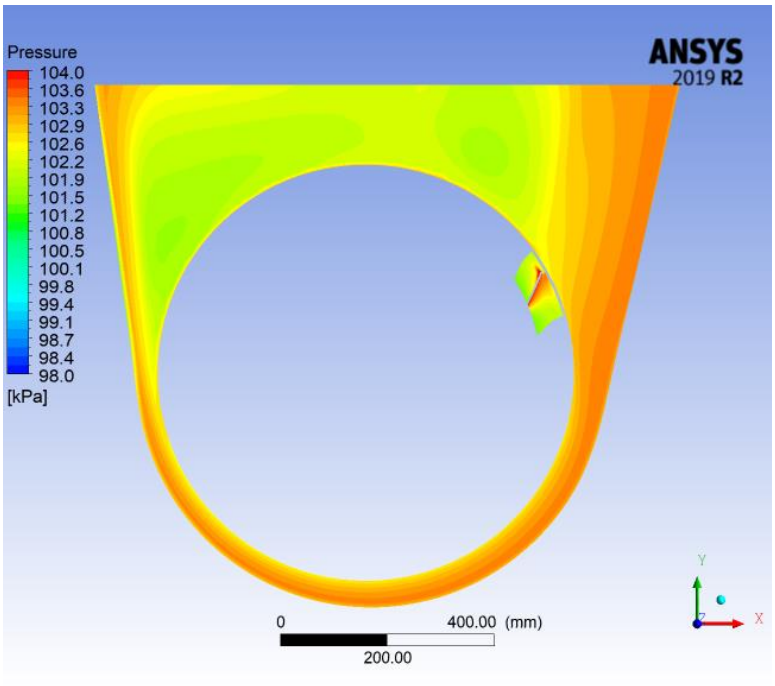
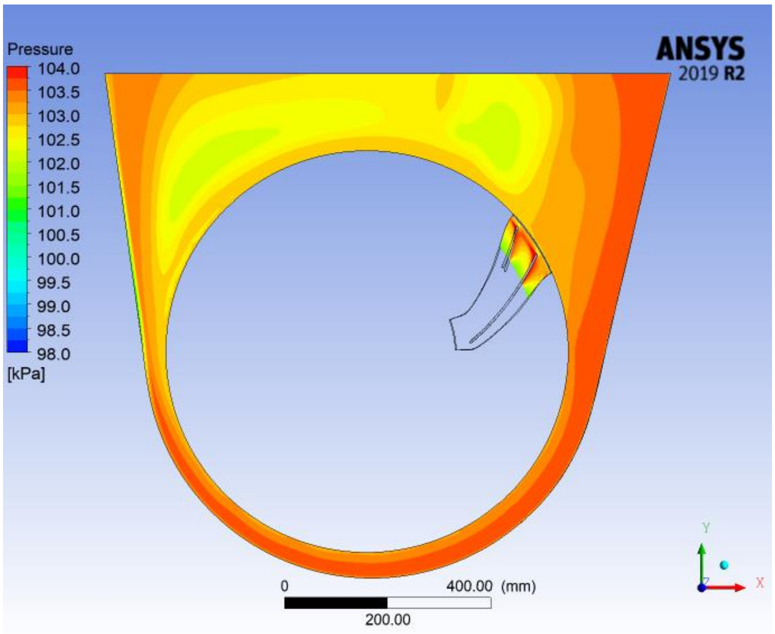
***Figure 6.9:** Static Pressure Plot at 1400RPM*

6.5 Pressure Contours

As soon as rotating impeller blade moves large volume of air through the volute casing, pressure distribution is produced. Brief comparison of pressure contours between modified geometry vs reference geometry is as following:

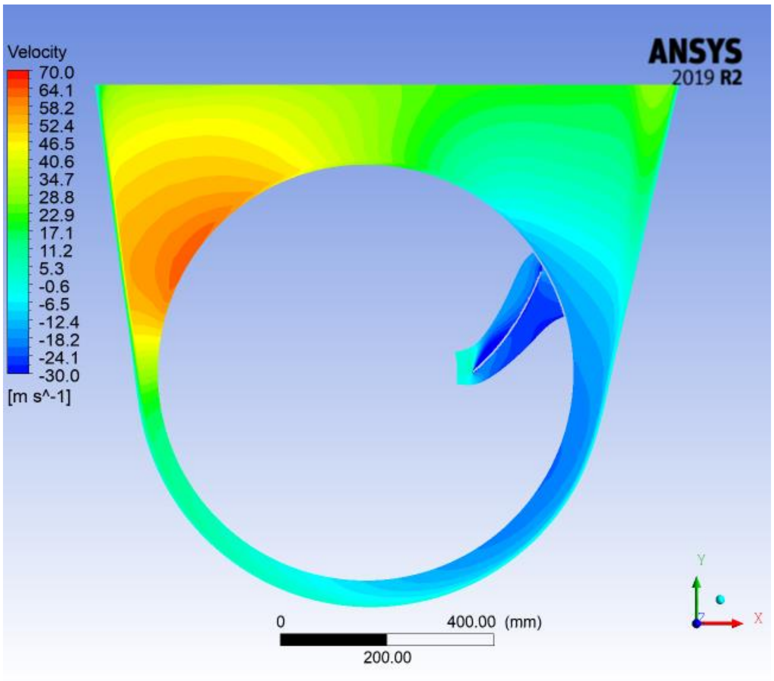
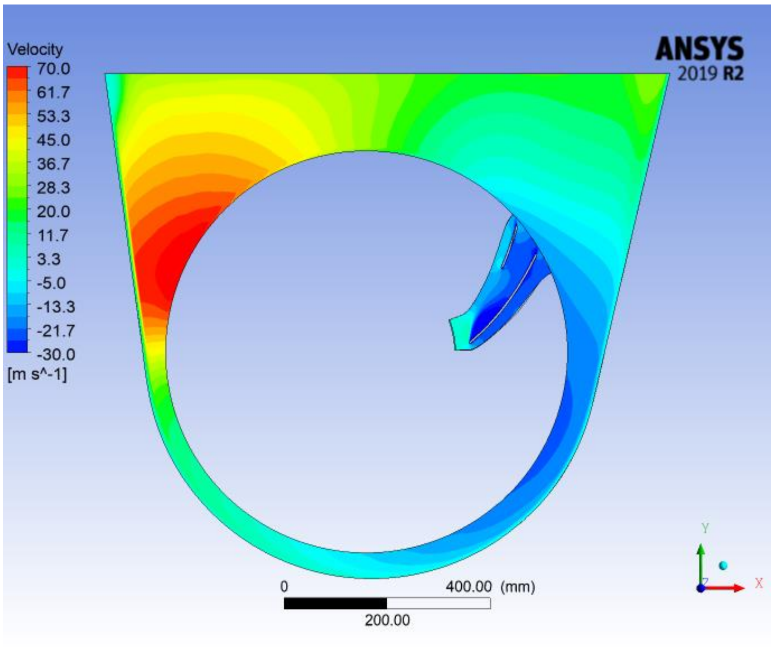
Sr	Pressure Contours	Details
a	 **Reference Geometry**	9mm Span 1400RPM Pressure Contours
	 **Modified Geometry**	

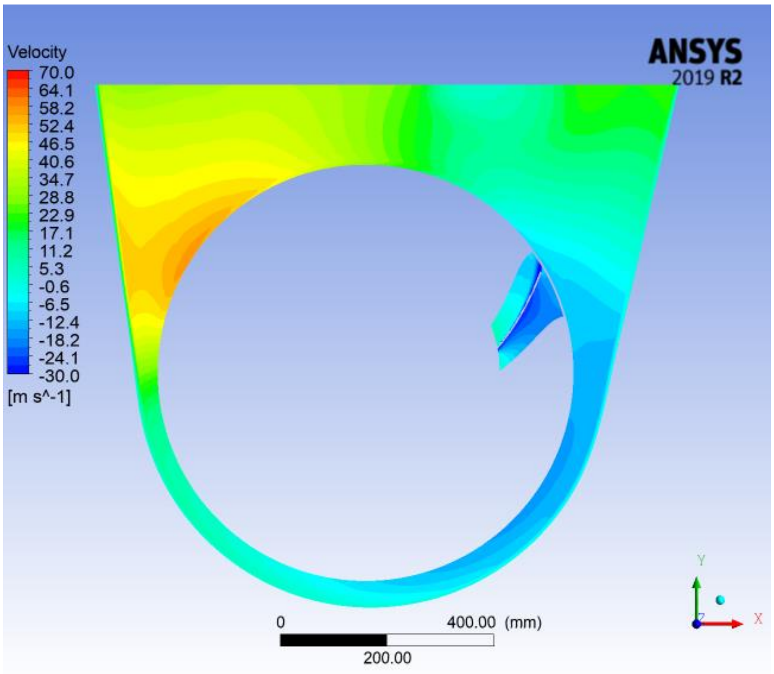
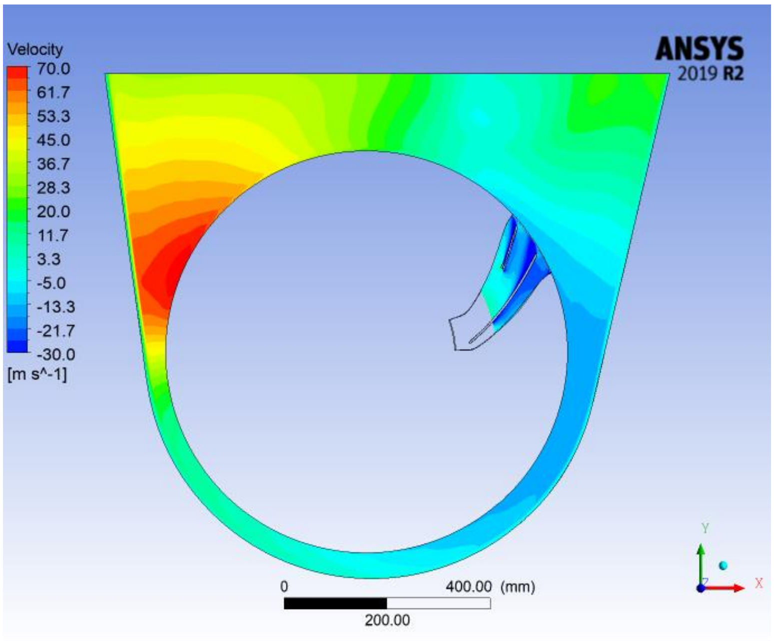
Sr	Pressure Contours	Details
c	 **Refrence Geometry**	90mm Span 1400RPM Pressure Contours
	 **Modified Geometry**	

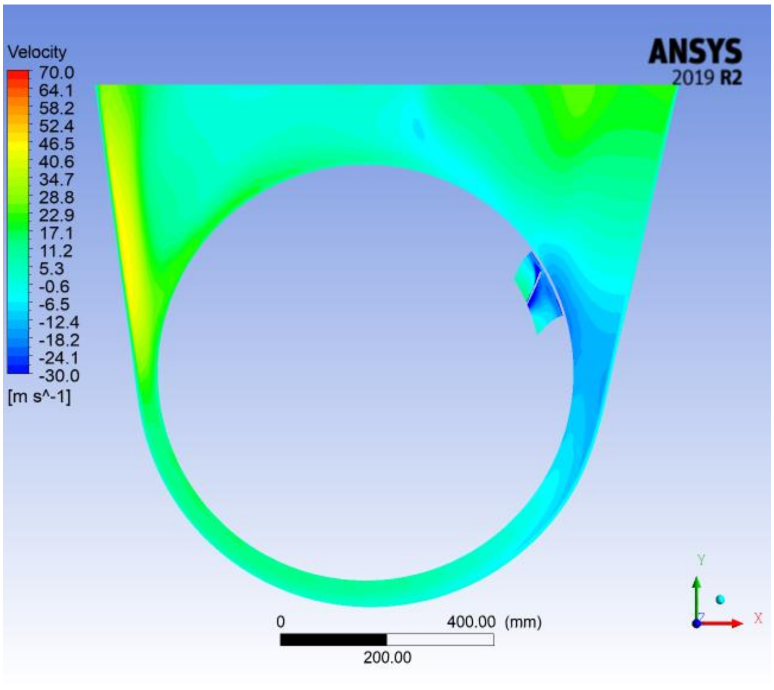
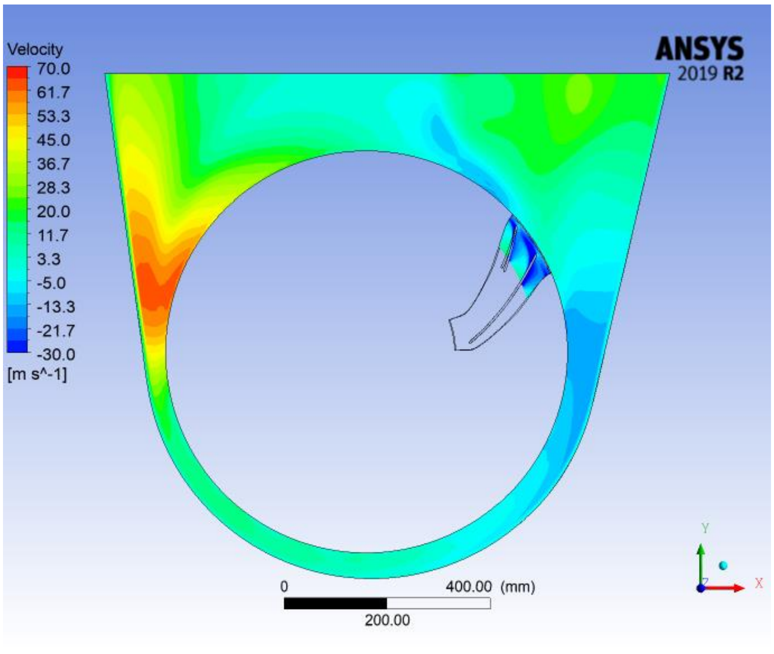
Sr	Pressure Contours	Details
e	 **Refrence Goemetry**	150mm Span 1400RPM Pressure Contours
	 **Modified Geometry**	

6.6 Velocity Contours

A rotating impeller blade is designed such that it throws the fluid radially outwards from the center. As soon as large volume is moved towards the volute casing of the fan, pressure head is generated. Following table shows velocity contours of fluid at different plane spans mentioned against each. Velocity contours shows fluid behavior in modified geometry vs reference geometry.

Sr	Velocity Contours	Details
a	 **Reference Geometry**	9mm Span 1400RPM Velocity Contours
	 Modified Geometry	

Sr	Velocity Contours	Details
c	 **Refrence Geometry**	90mm Span 1400RPM Velocity Contours
	 **Modified Geometry**	

Sr	Velocity Contours	Details
e	 **Refrence Geometry**	150mm Span 1400RPM Velocity Contours
	 **Modified Geometry**	

CHAPTER SEVEN

7 Conclusions and Future Recommendations

The subject research is performed on a backward curved centrifugal fan. It is fitted in a military vehicle, and serves the purpose of cooling an engine. This research is performed at College of Electrical and Mechanical, NUST. The study has been completed utilizing all available resources for both, experimentation and simulations. This research enabled us to investigate the fluid flow characteristics in a centrifugal fan. This section concludes the analysis and conclusion based on the data obtained after experiments and simulations. Recommendations are presented for improving the overall performance as an objective.

7.1 Conclusions

The behavior of backward curved centrifugal fan is thoroughly analyzed. At different design points, results have been recorded. Simulations have been performed at steady state conditions. Major conclusions drawn base on present research are appended below:

- Reference geometry has no significant effect based on varying number of impeller blades. This reflects, number of blades are already optimal.
- Splitter vanes significantly enhances the overall efficiency and static pressure of centrifugal fan.
- There are significant variations in the region surrounding impeller blade. This is because fan is major energy transferring unit.
- Length of splitter vanes have considerable effect on fan performance. An optimal geometry can be obtained by modifying the length of a splitter vanes.
- The is insignificant effect on the fan performance of a centrifugal fan, due to the angle between main blade and splitter vanes.

7.2 Future Recommendation

It is recommended to experimentally test the optimal geometry. Lab test of optimal geometry will strengthen the research. It will also enable endless possibilities of optimization in related assemblies.

Major focus in this research is the effect of splitter vanes. However, future projects can be done on optimization of volute casing. Effects of airfoil in a centrifugal fan can be evaluated for better performance. Weight reduction of cooling system is also a matter of significance.

Cooling system of a vehicle is constituted by a variety of assemblies. The overall optimization of a cooling system can be obtained if each assembly is individually optimized as well. Each component of a cooling system shall be optimized.

References

[1] Davood Khoeini, Mohammad Reza Tavakoli: The Optimum Position of Impeller Splitter Blades of a Centrifugal Pump Equipped with Vane Diffuser. FME Transactions (2018) 46, 205-210.

[2] Sabareesh R, Dr. G. Karuppusami, Mr. R. Mascomani: Effect of Splitter Vanes on Blower Performance. IJRDO-Journal of Mechanical and Civil Engineering. Volume-1 | Issue-4 | April,2015 | Paper-11.

[3] Chen-Kang Huang, Mu-En Hsieh: Performance Analysis and Optimized Design of Backward-Curved Airfoil Centrifugal Blower. HVAC&R Research. Volume 15, Number 3, May 2009.

[4] Man-Woong Heo, Jin-Hyuk Kim and Kwang-Yong Kim: Design Optimization of a Centrifugal Fan with Splitter Blades. Int. J. Turbo Jet-Engines 2015; 32(2): 143–154.

[5] Girish A R, Prajwal Sandyal Lokesh.K. S, Varun N: CFD Simulation of Centrifugal Blower Using ANSYS CFX. National Conference on Advances in Mechanical Engineering Science (NCAMES-2016).

[6] Jiří Polanský, and Roman Gášpár: Aerodynamic Performance Prediction of Centrifugal Fan with Forward Curved Blades. Transaction on Control and Mechanical Systems, Vol. 2, No. 6, pp. 267-271, Jun, 2013.

[7] Man-Woong Heo, Kyung-Hun Cha, Jin-Hyuk Kim, Kwang-Yong Kim: Parametric Study on Aerodynamic Performance of a Centrifugal Fan with Additionally Installed Splitter Blades. Proceedings of the ASME 2013; Fluids Engineering Division Summer Meeting FEDSM2013 July 7-11, 2013, Incline Village, Nevada, USA.

[8] N. Yagnesh Sharma1 and K. Vasudeva Karanth: Numerical Analysis of a Centrifugal Fan for Improved Performance Using Splitter Vanes. World Academy of Science, Engineering and Technology International Journal of Mechanical and Mechatronics Engineering Vol:3, No:12, 2009.

[9] Ruizi Zhang, Kaibin Wang, Yunlong Li, and Jingyin Li: Aerodynamic Optimization of Squirrel-Cage Fan with Dual Inlet. International Journal of Fluid Machinery and Systems. Vol. 11, No. 3, July-September 2018.

[10] Xiaomin Liu, Qun Dang and Guang Xi. Performance Improvement of Centrifugal Fan by Using CFD. Engineering Applications of Computational Fluid Mechanics Vol. 2, No. 2, Pp. 130–140 (2008).

[11] Jack B. EVANS, PE: Fan Selection and sizing to reduce inefficiency and frequency noise generation. JEACOUSTICS, Engineered Vibration Acoustic and Noise Solutions 1705 West Koeing Lanr, Austin, Texas 78756, USA, 2003.

[12] Madhwesh N., Vasudeva Karanth and Yagnesh Sharma: Impeller Treatment for a Centrifugal Fan using Splitter Vanes- A CFD Approach. Proceddings of the World Congress on Engineering 2011 Vol III WCE 2011, July 6-8, 2011, London, U.K.

[13] Jianfeng Yu, Jianming Qian: Pneumatic Handbook (Eighth Edition), 1997. Section: 2.9.1 Centrifugal Fans.