OPEN ACCESS INTERNATIONAL JOURNAL OF SCIENCE & ENGINEERING

MODELING AND STRUCTURAL ANALYSIS OF COMPOSITE CENTRIFUGAL BLOWER USING FEM

[1]M.Tech student, Department of Mechanical Engineering, AKULA SREE RAMULU COLLEGE OF ENGINEERING , TETALI, TANUKU, Andhra Pradesh
[2]Associate Professor, Department of Mechanical Engineering, AKULA SREE RAMULU COLLEGE OF ENGINEERING , TETALI,TANUKU, Andhra Pradesh
[1]GUBBALA SAIBABU[2] VADLAMUDI RAVIKUMAR
glrgsb@gmail.com Vadlamudi717@gmail.com

Abstract: A centrifugal blower is a mechanical device for moving air or other gases. The terms "blower" and "squirrel cage fan" (because it looks like a hamster wheel) are frequently used as synonyms. Rotating impellers increase the speedof the air blowing from otherend. Centrifugal blowers are used in naval applications and motors. The Contemporary blades in Centrifugal Blower used in naval applications are made up of Aluminum or Steel now we replaced the composite material (carbonfiber).
In this thesis the centrifugal blower modeling in CREO parametric software and analysis in ANSYS software with different materials in static analysis and different velocities in CFD analysis to find the fluidflow.
In this thesis the static analysis to determine the stress, deformation and strain with different materials (aluminum alloy, graphite and carbon fiber)
CFD analysis to determine the pressure drop, velocity, heat transfer coefficient and mass flow rate at different velocities (14, 16, 18, 20 and22m/s).

I INTRODUCTION

1.1 INTRDOUCTION

A centrifugal fan is a mechanical device for moving air or other gases. The terms "blower" and "squirrel cage fan", (because it looks like a hamster wheel), are frequently used as synonyms. These fans increase the speed and volume of an air stream with the rotatingimpellers. Centrifugal fans use the kinetic energy of the impellers to increase the volume of the air stream, which in turn moves them against the resistance caused by ducts, dampers and other components. Centrifugal fans displaceair radially, changing the direction (typically by 90°) of the airflow. They are sturdy, quiet, reliable, and capable of operating over a wide range of conditions.

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.

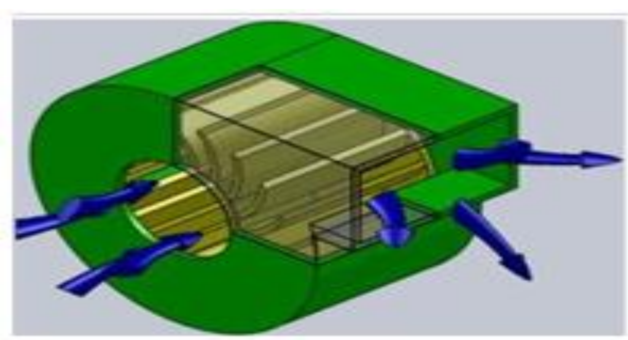

Figure 1:CENTRI FUGAL FAN

1.1.1 A CENTRIFUGALFAN

Centrifugal fans are not positive displacement devices. Centrifugal fans have certain advantages and disadvantages when contrasted with positive-displacement blowers.

The centrifugal fan is one of the most widely used fans. 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 sys tem for buildings. They are also well-suited for industrial processes and air pollution control systems.

The centrifugal fan is a drum shape composed of a number of fan blades mounted around a hub. As shown in the animated figure, the hub turns on a driveshaft mounted in bearings in 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.

1.2 HISTORY

The earliest mention of centrifugal fans was in 1556 by Georg Pawer (Latin: Georgius Agricola) in his book De Re Metallica, where he shows how such fans were used to ventilate mines. Thereafter, centrifugal fans gradually fell into disuse. It wasn't until the early decades of the nineteenth century that interest in centrifugal fans revived. In 1815 the Marquis de Chabannesadvocated the use of a centrifugal fan and took out a British patent in the same year. In 1827, Edwin A. Stevens of Bordentown, New Jersey, installed a fan for blowing air into the boilers of the steamship North America. Similarly, in 1832, the Swedish-American engineer John Ericsson used a centrifugal fan as blower on the steamship Corsair. A centrifugal fan was invented by Russian military engineer Alexander Sablukovin 1832, and was used both in the Russian light industry (such as sugar making) andabroad.

1.2.1CONSTRUCTION

The property that distinguishes a centrifugal fan from a blower is the pressure ratio it can achieve. In general, a blower can produce a higher pressure ratio. As per American Society of Mechanical Engineers (ASME) the specific ratio - the ratio of the discharge pressure over the suction pressure – is used for defining the fans and blowers (Table1).

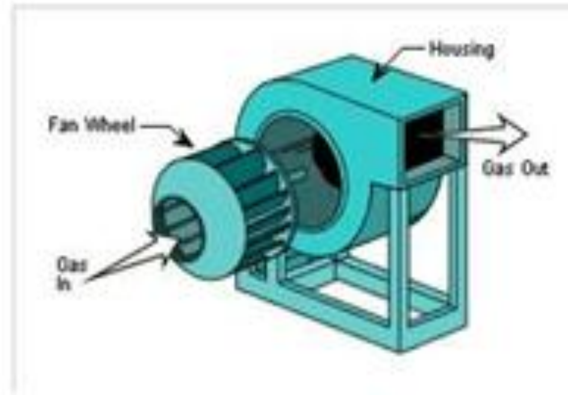

Figure 2: Components of a centrifugal fan

Main parts of a centrifugal fan are:

- Fanhousing
- Impellers
- Inlet and outletducts
- Drive shaft

1.2.2 FAN DAMPERS AND VANES

Fan dampers are used to control gas flow into and out of the centrifugal fan. They may be installed on the inlet side or on the outlet side of the fan, or both. Dampers on the outlet side impose a flow resistance that is used to control gas flow. Dampers on the inlet side (inlet vanes) are designed to control gas flow by changing the amount of gas or air admitted to the faninlet.

Inlet dampers (inletvanes) reduce fan energy usage due to their ability to affect the airflow pattern into the fan.

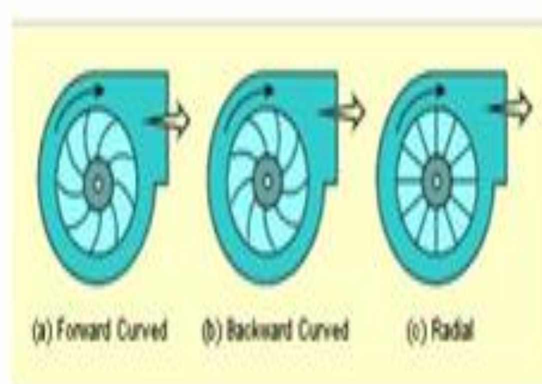

Figure 3: Centrifugal fan blades

1.3TYPES OF BLOWERS

1.3.1 CENTRIFUGALBLOWERS

Centrifugal Blower Centrifugal blowers use high speed impellers or blades to impart velocity to air or other gases. They can be single or multi-stage units. Like fans, centrifugal blowers offera number of blade orientations, including backward curved, forward curved, and radial. Blowers can be multi- or variable speed units. They are usually driven by electric motors, often through a belt and sheave arrangement, but some centrifugal blowers are directly coupled to drive motors.

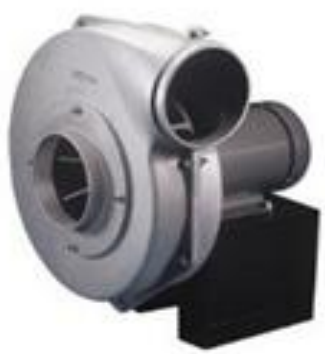

Figure 2 Centrifugal Blowers Positive

II LITERATURE REVIEW

Static and Dynamic Analysis of a Centrifugal Blower Using FeaVeeranjaneyuluItha1, T.B.S.Rao2,InternationalJournalof Engineering Research & Technology(IJERT)Vol.1Issue 8, October - 2012 ISSN: 2278-0181, pp. 1-11. 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.

Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers AtrePranavC. and ThundilKaruppaRaj R. School of Mechanical and Building Sciences, VIT University, Vellore-632014, Tamilnadu, INDIA. In this project work this paper is used to know how Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade impellers help to improve the efficiency of blades &optimize the weight. Fans are one of the types of turbo machinery which are used to move air continuously with in slight increase in static pressure.

.The performance of the fan system may range from free air to several cfm (cubic feet per min.). Selection of fan system depends on various conditions such as airflow rates, temperature of air, pressures, airstream properties, etc. Although,the fan is usually selected for nontechnical reasons like price, delivery, availability of space, packaging etc. The fan is always analyzed by its performance curves which are defined as the plot of developed pressure and power required over a range of fan generated air flow. Also these fan characteristic curves can be used to data like fan bhp for selection of the motor being used. The centrifugal fans with impellers having blades of Airfoil section are considered as the high efficiency impellers among the six types Airfoil blades, Backward Inclined single thickness blades, Backward curved blades, forward curved blades, radial tip blades and radial blades. The present study gives the design methodology for these high efficiency impellers which include the numerical design procedure and the CFD analysis of it. The CFD part is used for improvement the results of Static Pressure generated at the entry to the impeller, static efficiency. The CFD optimizational so helped to improve the flow pattern through the centrifugal fan system.

A numerical Study on the Acoustic Characteristics of a Centrifugal Impeller with a Splitter Wan-Ho Jeon1 1 Technical Research Lab., CEDIC Ltd., #1013, Byuksan Digital Valley II,Kasan- dong. This paper is used to know Acoustic Characteristics of a Centrifugal Impeller with a Splitter. Centrifugal turbo machines are commonly used in many air-moving devices due to their ability to achieve relatively high-pressure ratios in a compact configuration compared with axial fans. They are often found in gas turbine engines, heating ventilation and air conditioning systems, and hydraulic pumps.

Because of their widespread use, the noise generated by these machines often causes serious environmental issues. The turbo machinery noise is often dominated by tones at blade passage frequency and its higher harmonics. This is mainly due to strong interactions between the flow discharged from the impeller and the cut off of the casing. In addition to discrete tones, the broadband noise is also generated due to the separation, turbulence mixing, and the vortex interaction process. The numerical method to predict the flow- and acoustic-fields of an axial fan have been studied by many researchers. On the contrary, the numerical prediction method for the centrifugal fan has not been studied widely. This is due to the difficulty in obtaining detailed information of flow-fields and implementing scattering effects by the casing. A numerical method to analyze the acoustic field of the centrifugal Fan was developed recently by Jeon and Lee. This method predicts the acoustic pressure with an accuracy of maximum error of 2dB, when compared with the measured data.

Evaluation of Static &Dynamic Analysis of a CentrifugalBlower Using FeaMohdJubairNizami, Ramavath Sunman, M.GuruBramhanandaReddy, International Journal Of Advanced Trends in Computer Science and Engineering, Vol.2, Issue 7, January-2013, pp. - 316-321. To study static and dynamic analysis of blower so as to reduce vibrations &impact. Centrifugal blowers are used extensively for onboard naval applications have high noiselevels.The noise produced by a rotating component is mainly due to random loading force on the blades and periodic iteration of incoming are with the blades of the rotor. The contemporary blades in naval applications are made up of aluminum or steel and generate noise that causes disturbance tothe peopleworking near the blower. 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 usingHYPERMESH10. 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 Aluminum and composite blower to find out first 10 natural frequencies.

Numerical Analysis of Internal Flow Field of Multi-Blade Centrifugal Fan for Floor Standing Air-Conditioner Jia Bing Wang Huazhong University of Science and Technology. In this project work this paper is used to Numerical Analysis of Internal Flow Field of Multi-Blade Centrifugal Fan for Floor Standing Air-Conditioners oas to improve discharge of the blower. The flow field in a centrifugal fan is highly complex with flow reversal taking place on the suction side of impeller and diffuser vanes. Generally performance of the centrifugal fan could be enhanced by judiciously introducing splitter vanes so as to improve the diffusion process. An extensive numerical whole field analysis on the effect of splitter vanes placed in discrete regions of suspected separation points is possible using CFD. This paper examines the effect of splitter vanes corresponding to various geometrical locations on the impeller and diffuser. The analysis shows that the splitter vanes located near the diffuser exit improves the static pressure recovery across the diffusing domain to a larger extent. Also it is found that splitter vanes located at the impeller trailing edge and diffuser leading edge at the mid-span of the circumferential distance between the blades show a marginal improvement in the static pressure recovery across the fan. However, splitters provided near to the suction side of the impeller trailing edge (25% of the circumferential gap between the impeller blades towards the suction side), adversely affect the static pressure recovery of the fan.

III INTRODUCTION TO CAD

Computer-aided design (**CAD**) is the use of computer systems (or workstations)to aid in the creation, modification, analysis, or optimization of a design.CAD software is used to increasethe productivity of the designer, improve the quality of design, improve communications through documentation, and to create a database for manufacturing. CAD output is often in the form of electronic files for print, machining, or other manufacturing operations. The term **CADD** (for Computer Aided Design and Drafting) is alsoused.

Its use in designing electronic systems is known as electronic design automation,or **EDA**. In mechanical design it is known as mechanical design automation (**MDA**) or **computer-aided drafting** (**CAD**), which includes the process of creating a technical drawing with the use of computer software.

CAD software for mechanical design uses either vector-based graphics to depict the objects of traditional drafting, or may also produce raster graphics showing the overall appearanceofdesignedobjects.However, it involvesmore than just shapes. As in the manual drafting of technical and engineering drawings, the output of CAD must convey information, such as materials,processes,dimensions,and tolerances, according to application- specificconventions.

CAD may be used to design curves and figures in two-dimensional (2D) space; or curves, surfaces, and solids in three-dimensional (3D) space.

CAD is an important industrial art extensively used in many applications, including automotive, shipbuilding,and aerospace industries, industrial and architecturaldesign,prosthetics, and many more. CAD is also widely used to produce computeranimation for special effects in movies, advertising and technical manuals, often called DCC digital content creation.The modern ubiquity and power of computers means that even perfume bottles and shampoo dispensers are designed using techniques unheard of by engineers of the 1960s. Because of its enormous economic importance, CAD has been a major driving force for research in computational geometry,computer graphics (both hardware and software), and discrete differentialgeometry.

3.1 INTRODUCTION TO CREO

CREO is a one of the world's leading high-end CAD/CAM/CAE software packages. CREO (Computer Aided Three dimensional Interactive Application) is a multi-platform PLM/CAD/CAM/CAE commercial software suite developed by Dassault Systems and marketed world-wide by IBM.CREO is written in the C++ programming language. CREO provides open development architecture through the use of interfaces, which can be used to customize or develop applications. The application programming interfaces supported Visual Basic and C++ programming languages. Commonly referred to as 3D Product Lifecycle Management (PLM) software suite, CREO supports multiple stages of product development. The stages range from conceptualization, through design (CAD) and manufacturing (CAM),until analysis (CAE). Each workbench of CREO V5 refers an each stage of product development for different products. CREO V5 features a parametricsolid/surface-based package which uses NURBS as the core surface representation and has several workbenches that provide KBE (Knowledge Based Engineering)support.

Feature-based Modeling:

In CREO V5, solid models are created by integrating a number of building blocks called features.

Parametric modelling:

The parametric nature of a software package is defined as its ability to use the standard properties or parameters in defining the shape and size of a geometry.

Associatively:

Associatively ensures that if any modification is made in the model in any one of the workbenches of CREO V5, it is automatically reflected in the other work benches immediately.

Its use in designing electronic systems is known as electronic design automation, or **EDA**. In mechanical design it is known as mechanical design automation (**MDA**) or **computer-aided drafting** (**CAD**), which includes the process of creating a technical drawing with the use of computer software.

B-Rep modelling: Most of the components Designed using CREO V5 are based on B-Rep modelling technique i.e. models are created by extruding the boundary of the model in a specified direction.

3.2 ADVANTAGES OF CREO PARAMETRICSOFTWARE

1. Optimized for model-based enterprises
2. Increased engineer productivity
3. Better enabled concept design
4. Increased engineering capabilities
5. Increased manufacturing capabilities
6. Better simulation
7. Design capabilities for additive manufacturing

IV INTRODUCTION TO FEA

4.1INTRODUCTION TO FEA

Finite Element Analysis(FEA)was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus to obtain approximate solutions to vibration systems. Shortly thereafter, a paper published in 1956 by M.

Turner, R. W. Clough, H. C. Martin, and L. J. Top established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

By the early 70's, FEA was limited to expensive mainframe computers generally owned by the aeronautics, automotive, defense, and nuclear industries. Since the rapid decline in the cost of computers and the phenomenal increase in computing power, FEA has been developed to an incredible precision. Present day supercomputers are now able to produce accurate results for all kinds ofparameters.

FEA consists of a computer model of a material or design that is stressed and analyzed for specific results. It is used in new product design, and existing product refinement. A company is able to verify a proposed design will be able to perform to the client's specifications prior to manufacturing or construction. Modifying an existing product or structure is utilized to qualify the product or structure for a new service condition. In case of structural failure, FEA may be used to help determine the design modifications to meet the new condition.

There are generally two types of analysis that are used in industry: 2-D modeling, and 3- D modeling. While 2-D modeling conserves simplicity and allows the analysis to be run on a relatively normal computer, it tends to yield less accurate results. 3-D modeling, however, produces more accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modeling schemes, the programmer can insert numerous algorithms(functions) which may make the system behave linearly or non-linearly. Linear systems are far less complex and generally do not take into account plastic deformation. Non- linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

4.2RESULTS OF FINITE ELEMENT ANALYSIS

Pre-processing: The user constructs a model of the part to be analyzed in which the geometry is divided into a number of discrete sub regions, or elements," connected at discrete points called nodes." Certain of these nodes will have fixed displacements, and others will have prescribed loads.

These models can be extremely time consuming to prepare, and commercial codes vie with one another to have the most user-friendly graphical –pre-processor" to assist in this rather tedious chore. Some of these pre-processors can overlay a mesh on a pre-existing CAD file, so that finite element analysis can be done conveniently as part of the computerized drafting-and-design process.

V INTRODUCTION TOANSYS

ANSYS is general-purpose finite element analysis (FEA) software package. Finite Element Analysis’s numerical method of deconstructing a complex system into very small pieces (of user-designated size) called elements. The software implements equations that govern the behaviour of these elements and solves them all; creating a comprehensive explanation of how the system acts as a whole. These results then can be presented in tabulated, or graphical forms. This type of analysis is typically used for the design and optimization of a system far too complex to analyze by hand. Systems that may fit into this category are too complex due to their geometry, scale, or governing equations.

ANSYS is the standard FEA teaching tool within the Mechanical Engineering Department at many colleges. ANSYS is also used in Civil and Electrical Engineering, as well as the Physics and Chemistry departments.

ANSYS provides a cost-effective way to explore the performance of products or processes in a virtual environment. This type of product development is termed virtual prototyping.

5.1 SPECIFIC CAPABILITIES OF ANSYS Structural

Structural analysis is probably the most common application of the finite element method as it implies bridges and buildings, naval, aeronautical, and mechanical structures such as ship hulls, aircraft bodies, and machine housings, as well as mechanical componentssuch as pistons,machine parts, andtools.

- Static Analysis - Usedto determine displacements, stresses, etc. under static loading conditions. ANSYS can compute both linear and nonlinear static analyses. Nonlinearities can include plasticity, stress stiffening, large deflection, large strain, hyper elasticity, contact surfaces, and creep.

Transient Dynamic Analysis - Used to determine the response of a structure to arbitrarily time-varying loads. All nonlinearities mentioned under Static Analysis above are allowed.

- Buckling Analysis-Used to calculate the buckling load sand determine the buckling mode shape. Both linear (eigen value) buckling and nonlinear buckling analyses are possible.

5.2 THERMAL

ANSYS is capable of both steady state and transient analysis of any solid with thermal boundary conditions. Steady-state thermal analyses calculate the effects of steady thermal loads on a system or component. Users often perform a steady-state analysis before doing a transient thermal analysis, to help establish initial conditions. A steady-state analysis also can be the last step of a transient thermal analysis; performed after all transient effects have diminished. ANSYS can be used to determine temperatures, thermal gradients, heat flow rates, and heat fluxes in an object that are caused by thermal loads that do not vary over time

5.3 MODAL ANALYSIS

A modal analysis is typically used to determine the vibration characteristics (natural frequencies and mode shapes) of a structure or a machine component while it is being designed. It can also serve as a starting point for another, more detailed ,dynamic analysis, such as a harmonic response or full transient dynamic analysis.

Modal analyses, while being one of the most basic dynamic analysis types available in ANSYS, can also be more computationally time consuming than a typical static analysis. A reduced solver, utilizing automatically or manually selected master degrees of freedom is used to drastically reduce the problem size and solution time.

5.4 HARMONIC ANALYSIS

Used extensively by companies who produce rotating machinery, ANSYS Harmonic analysis is used to predict the sustained dynamic behavior of structures to consistent cyclic loading. Examples of rotating machines which produced or are subjected to harmonic loading are:

- **Turbines**

oGas Turbines for Aircraft and PowerGeneration,

oSteam Turbines,

oWind Turbine ,

oWaterTurbines,

oTurbopumps

•Internal Combustionengines

•Electric motors andgenerators

•Gas and fluidpumps

•Discdrives

A harmonic analysis can be used to verify whether or not a machine design will successfully overcome resonance, fatigue, and other harmful effects of forcedvibrations.

VI 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.

6.1 METHODOLOGY

In all of these approaches the same basic procedure is followed.

- During pre-processing
- The geometry(physical bounds) of the problem is defined.
- The volume occupied by the fluid is divided into discrete cells (the mesh). The mesh may be uniform or non-uniform.

The physical modeling is defined – for example, the equations of motion+ enthalpy + radiation + species conservation

Boundary conditions are defined. This involves specifying the fluid behavior and properties at the boundaries of the problem. For transient problems, the initial conditions are also defined.The simulation is started and the equations are solved iteratively as a steady-state or transient.

6.2 STATIC ANALYSIS CENTRIFUGAL BLOWER PROPELLER BLADE

6.2.1 MATERIAL -ALUMINUMALLOY

Save creo Model as .iges format

→→Ansys → Workbench→ Select analysis system → static structural → double click

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

→→ Select mesh on work bench → right click →edit

Double click on geometry → select MSBR → edit material →

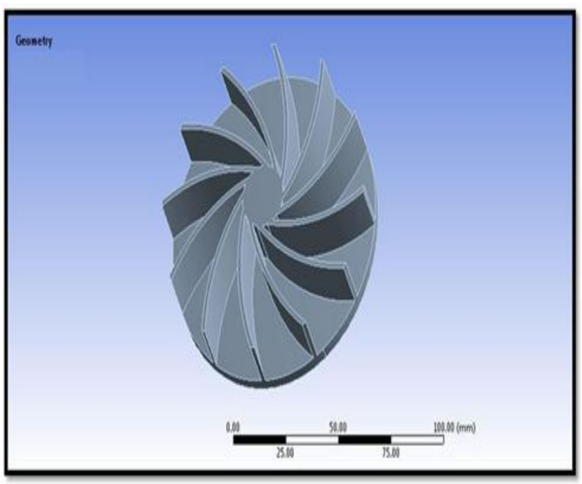

Figure 5:import geometry

Select mesh on left side part tree → right click → generate mesh →

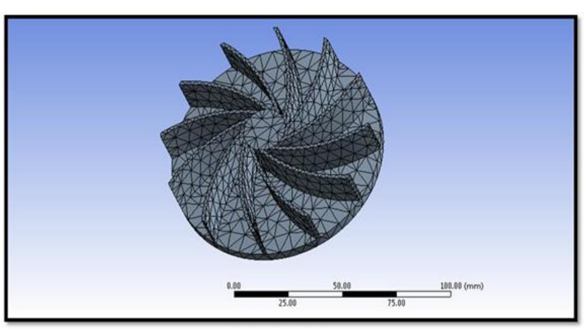

Figure 6 meshed geometry

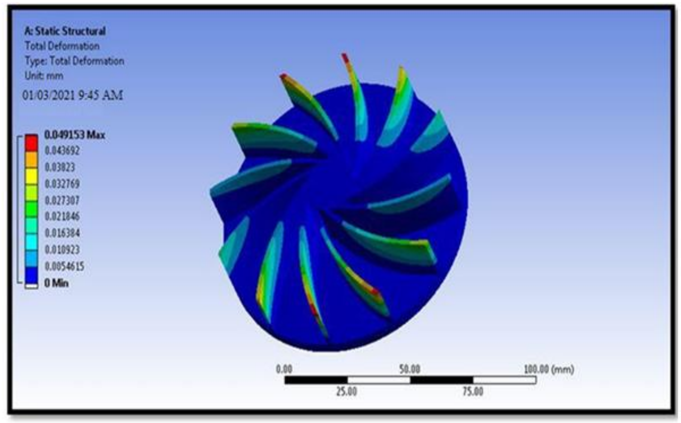

FIG7:DEFORMATION

6.2.1 DEFORMATION

6.2.2 STRESS

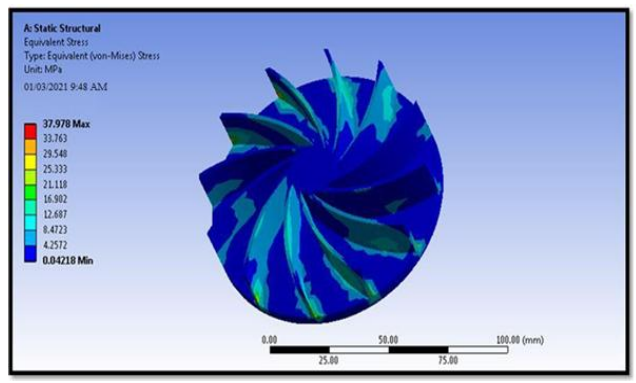

Figure 8 EQUIVALENT STRESS

6.2.3 STRAIN

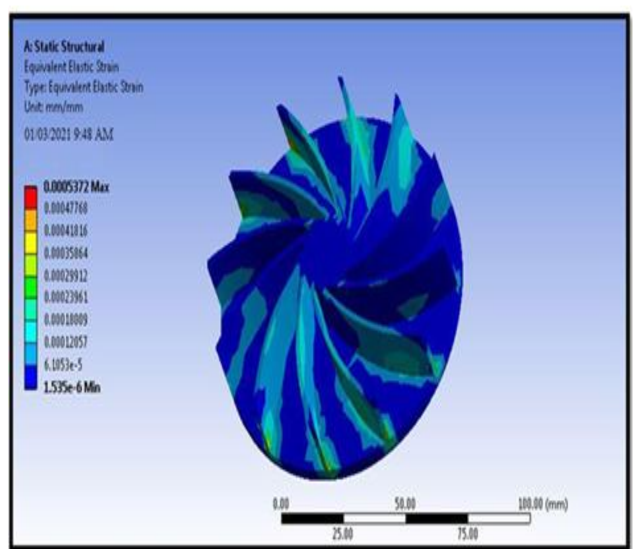

Figure 9: EQUIVALENT STRAIN

6.3 MATERIAL -GRAPHITE

6.3.1 DEFORMATION

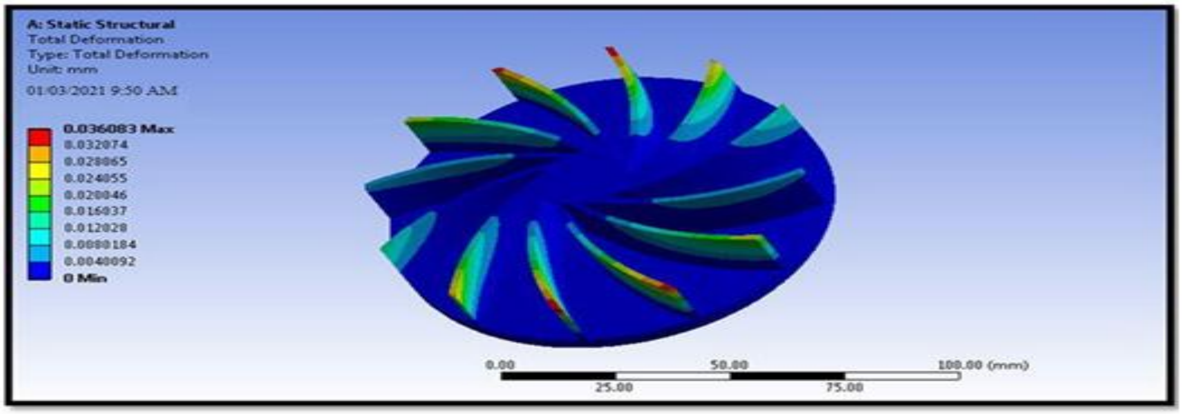

Figure 10 DEFORMATION

6.3.2 STRAIN

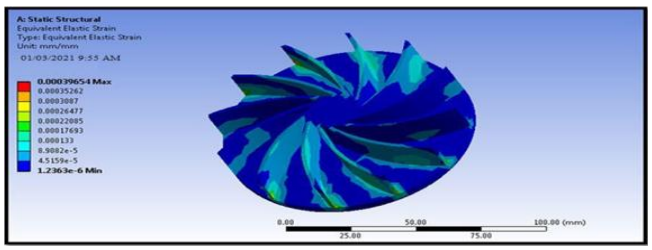

Figure 11STRAIN

6.4 MATERIAL -CARBN FIBER

6.4.1 DEFORMATION

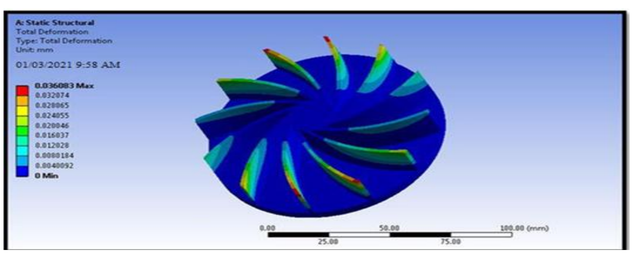

Figure 12:DEFORMATION

6.4.2 STRESS

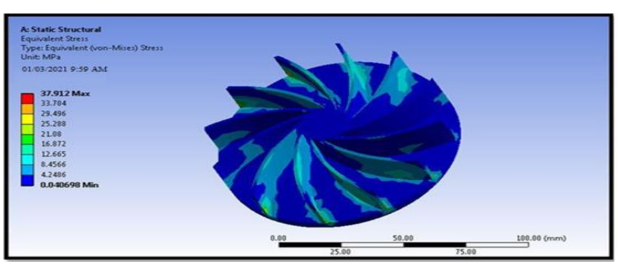

Figure 13STRESS

6.4.3 STRAIN

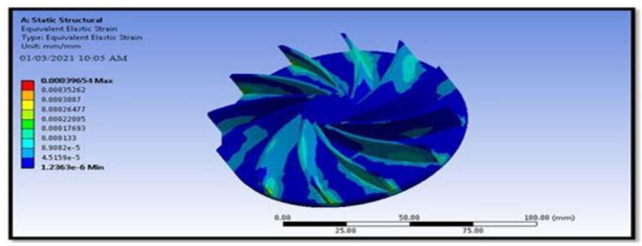

Figure 14:STRAIN

6.5 CFD ANALYSIS OF CENTRIFUGAL BLOWER ATVELOCITY-14m/s

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

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

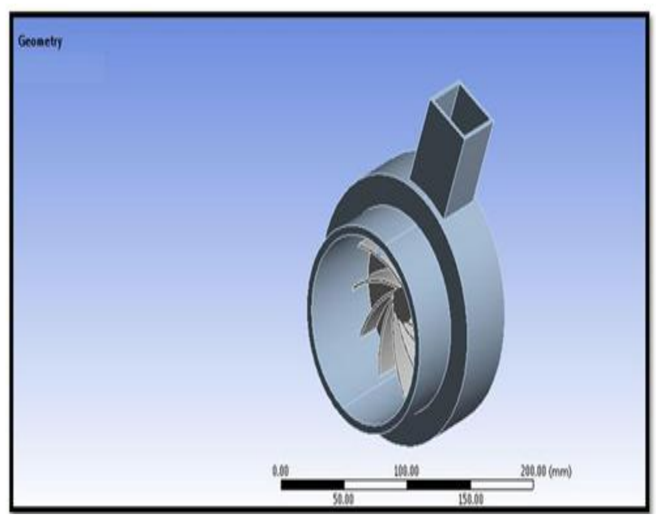

Figure 15: import geometry

→→ select mesh on work bench → right click

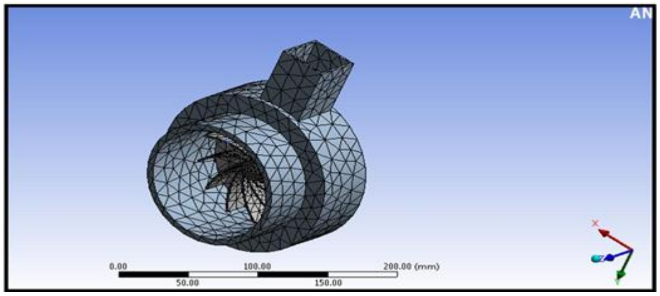

Figure 16 meshed geometry

6.5.1 PRESSURE

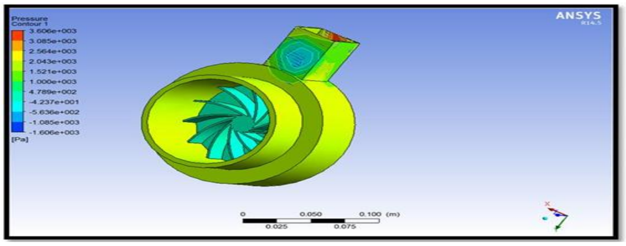

Figure 17PRESSURE

6.5.2 VELOCITY

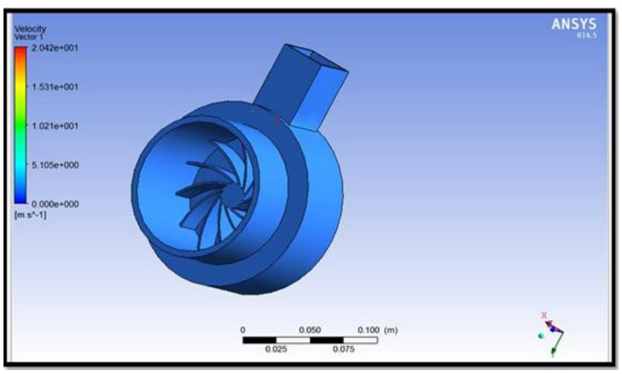

Figure 18 VELOCITY

6.5.3 HEAT TRANSFER COEFFICIENT

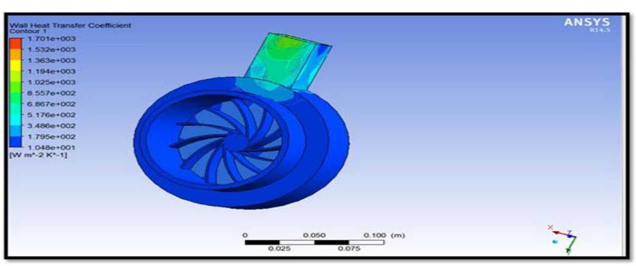

Figure 19:HEAT TRANSFER COEFFICIENT

7 RESULTS AND CONCLUSION

7.1 STATICANALYSISRESULTS

Material	Deformation (mm)	Stress (N/mm²)	Strain
Aluminum alloy	0.049153	37.978	0.0005372
Carbon fiber	0.036083	37.912	0.00039654
Graphite	0.0073159	37.966	7.94e-5

7.2 CFDANALYSISRESULTS

Inlet velocity(m/s)	Pressure (pa)	Heat transfer coeffiecent(w/m²k)	Velocity (m/s)	Mass flow rate(kg/sec)
14	3.606e+03	1.701e+03	2.04e+01	0.0049077
16	3.10e+03	1.158e+03	1.65e+01	0.00561029
18	4.73e+004	3.13e+03	9.703e+01	0.0063122
20	5.233e+03	1.036e+03	2.35e+01	0.007018
22	6.0523e+04	1.673e+03	3.166e+01	0.0077166

7.3 CONCLUSION

Centrifugal blowers are used in naval applications and motors. The Contemporary blades in Centrifugal Blower used in naval applications are made up of Aluminum or Steel now we replaced the composite material (carbonfiber).

In thisthesisthecentrifugalblowermodelinginCREO parametricsoftwareandanalysis in ANSYS software with different materials in static analysis and different velocities in CFD analysis to find the fluidflow.

In this thesis the static analysis to determine the stress, deformation and strain with different materials (aluminum alloy, graphite and carbonfiber).

By observing the static analysis the stress increases for carbon fiber composite material compare with aluminum alloy andgraphite.

By observing the CFD analysis the pressure drop, velocity, mass flow rate and heat transfer coefficient values are increases by increasing the inletvelocity.

REFERENCES

1.S.Rajendranand Dr.K.Purushothaman, –Analysis of a centrifugal pump impeler using ANSYS-CFX,‖ International Journal of Engineering Research&Technology,Vol.1,Issue3, 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.UshaShri ansC.Syamsundar, –computational analysis on performance of a centrifugal pump impel er,‖ 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. Mentzosand D.P. Margaris, –Parametric Study of a Centrifugal Pump Impel er by Varying the Outlet BladeAngle,‖ The Open Mechanical Engineering Journal, no 2, 75-83,2008.

5. Marco Antonio Rodrigues Cunhand Helcio Francisco Vila Nova, –Cavitation modelingof a centrifugal pump impel er,‖ 22nd International Congress of Mechanical Engineering, Ribeirao Petro, Sao Paulo, Brazil,2013.

6. Mohammed KhudhairAbbas, ―cavitation in centrifugal pumps,‖DiyalaJournal of Engineering Sciences, pp. 170- 180,2010.
7. Abdulkadir Aman, Sileshi Kore and Edessa Dribssa, ―Flow simulation and performance prediction of centrifugal pumps using cfd-tool,‖ Journal of EEA, Vol. 28,2011.
8 .Erik Dick, Jan Vierendeels, Sven Serbruynsand John VandeVoorde, ―Performance prediction of centrifugal pumps with cfd-tools,‖ Task Quarterly 5, no 4, 579–594,2001.
9. S. C. Chaudhari, C. O. Yadav and A. B. Damo, ―A comparative studyofmixflow pump impell er cfdanalysis and experimental data of submersible pump,‖ International Journal of Research in Engineering &Technology, Vol. 1, Issue 3, 57-64,2013.