INTERNATIONAL JOURNAL OF ENGINEERING MANAGEMENT SCIENCE
journal homepage: www.ijems.online/index.php/ijems/index

Design and Analysis of Blower Impeller

Md Aminuddin [1] ,Sk Darain Shahidi [2] ,Md Faisal Hussain [3] ,Md Sarwar [4] ,Shailandra Kumar Prasad [5]
[1, 2, 3, 4] Final year B. Tech. student, Mechanical Engineering Dept., RVS College of Engineering and Technology, Jamshedpur
[5] Assistant Professor, Mechanical Engineering Dept., RVS College of Engineering and Technology, Jamshedpur

How to Cite the Article: Md Aminuddin et. al., (2023). Design and Analysis of Blower Impeller. *International Journal of Engineering Management Science , 3(2), 1-7.*

Keyword
Blower, impeller, casing, design, model, analysis

Abstract
The primary objective of this study is to design and analysis of a blower impeller with the usage of many engineering methods and tools. This blower impeller is an important element of various commercial and industrial applications. Its efficiency and performance effects straight to the general procedure efficiently. In optimizing the overall system's performance, the blower impeller plays a significant role. The results of this study are used to detect the crucial areas in the impeller which need modification or demand further support. This study is successful in terms of designing and exploring a blower impeller that fulfills design demands and specifications which optimizes efficiency and performance. In general, this study reveals the value of using many engineering tools and methods in terms of analysis and design of blower impeller.

Introduction

In different industrial applications, centrifugal blowers have been used widely, which is efficient as long as the high-pressure rise and flow rate are moderate. There has main two parts of the blower, they are, impeller and casing. The impeller is usually assumed an essential part of the suction motor as its accommodation and engine are joined as a unit. The impeller caused by the blower post adds speed to the fluid composition by centrifugally casting the fluid from the impeller vane. The main idea here is that the energy which has been created here is kinetic energy. Quantity of power the delivered fluid coordinates the acceleration at the tip of the impeller or the tip of the vane. In the year 1995, Addison developed that the larger the impeller is or the faster it spins the higher the velocity of the fluid at the tip of the vane and the greater the force fluids are provided.

Aims and Objectives of Study

Aim

The current study is aimed at reporting on the design and developing blower impeller-specific analysis.

Objective

The objective of this study is to design a more satisfactory understanding of blower impellers and their influence on the performance of systems of mechanical. This study's objectives include

1. Designing a blower impeller that can enhance the performance and efficiency of the general system.

2. Examining the implementation of the blower impeller with the usage of FEA or Finite Element Analysis and CFD or Computational Fluid Dynamics techniques.
3. Comparing the analytical outcomes with the theoretical computations.
4. Optimizing the structure of the blower impeller established on the analysis.
5. Introducing the results and conclusions of this study to academics and industry communities.

Methodology

In Blower Design single-stage centrifugal blower is studied. From the industry of wheat mills, input data for design measures has been taken.

Design of data:
Rates of Airflow, Q= 1.2 $m3/s$
Speed of rotation, N=3800 rpm
The air force of inlet, Pa = 101.353 kPa

The structure of the centrifugal blower includes various Interdependent variables therefore their different possible designs for the same duty. To identify the flow rate, power input, and entire adiabatic head needed to be known. When the calculations of the theory have been completed, the very following stage is to develop a 2D and a 3D model of impeller (Baskar and Joseph 2022). With the usage of CAD software. This software permitted the outcome of the virtual and detailed model of the impeller, which could be adjusted and viewed among three dimensions. This model involves all important aspects of the impeller, like the hub, blades, and shroud.

The creation of the 3D and a 2D model of the impeller permitted the team to imagine persuasively and do whatever is required to change, to make sure that it satisfies the specifications of the design. To enhance the performance of the impeller, design the team utilizes a 3D model.

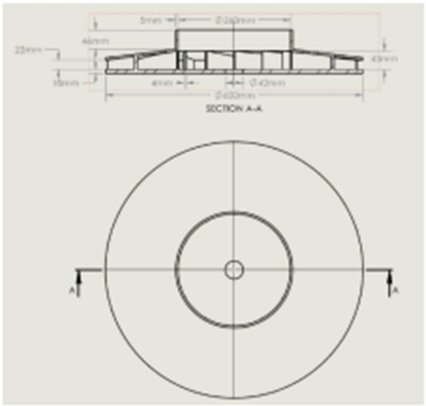

Fig 1: top and front view of the impeller

The CAD software which is used to create the 3D model, is also essential in the process of design. The software permitted 3D model manipulation to analyze and test the implementation of the impeller underneath different configuration choices and different circumstances (Li and Zhang 2022). The team conducts FEA or finite element analysis, by the software among impeller designs to identify their structural probity. Therefore, this chapter gives a clear definition of the design procedure and actions taken to develop a 3D replica of the impeller.

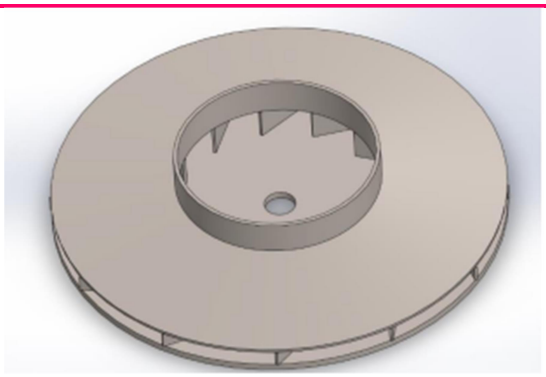

Fig 2: a 3D model of the impeller

In this study, various models and techniques have been used.
For Finite Element Analysis,

I. Analysis of static structure: this is a software tool that is used to analyze the problems of structural mechanics (Stoyanov and Nikolov 2022). Engineers can analyze the manners of the installation under various loading situations, like thermal limitations, fluid limitations, and mechanical limitations.
II. Modal analysis: this is a mechanism of software that is used to study the structures. To study the Predicts the dynamic behavior of design and their natural commonness's, size, modes, and corresponding damping proportion.
III. Analysis of transient thermal: this is used to impersonate the change of the temperature in a system over a period of time. It includes cracking a bunch of equations that define the procedure of heat transfer that took place in a system, like convention, conduction, and radiation.

For Computational Fluid Dynamics analysis

I. Vista CPD: this software is used to forge the phenomena of heat transfer and liquid flow. Vista CPD uses a limited volume and provides a range of methods and solvers for discretizing the governing problem choices for efficiency measures.
II. Turbo Grid: it is a technical software that is used for high-quality, structured creation forgings for turbo machinery applications.
III. BladeGen: this is used for the design of the blades of turbo machinery. The user can develop blades based on diverse input parameters, like conditions of the flow, blade geometry, and requirements of the performance.
IV. Volute Modal: this process includes the creation of a 3D model of casing volute, linking it, involving border states, cracking the equations, and processing the results to identify the modes of vibration.
V. CFX: it is used for emulating the manners of gases, liquids, and heat transfer among a broad range of applications in engineering.

Data Analysis and Findings
Analysis directs to the performance and feature evaluation procedure impeller design using modeling strategies and computer simulation. Odyjas et al. (2023) stated that, the purpose of the study is to forecast how the impeller will behave under various operational states, to determine possible issues with the design, and make changes to enhance its efficiency and performance. There are two types of analysis that are conducted in this study.

a) FEA or Finite Element Analysis
b) CFD or Computational Fluid Dynamics analysis

Finite element analysis (FEA)
Data of engineering
This indicates data that engineers have gathered, studied, and utilized, to create, develop, trial, and optimize outcomes, techniques, and procedures (Ren et al. 2022). It may take various forms, involving technical drawings, metrics of performance, specifications, and numerical data. Some typical types of engineering data are:

1. Data of performance: this involves data based on performance on methods, procedures, and products. Like efficiency, precision, and acceleration. To optimize configurations, detect rooms for improvement, and confirm that goods meet the demands of performance.
2. Data of test: this involves the collection of data during the validation and testing procedure. Like, tests of fatigue, stress, and implementation.
3. Data of design: this incorporates pictures, schematics, and models which explain the dimensions, geometry, and product components and techniques.
4. Data of manufacture: (Ren et al. 2022) this is manufacturing connected data, like, tolerances, material properties, and rates of production.
5. Data of simulation: this involves data developed when computer simulation of procedures and products is ongoing, like, FEA and CFD. This data is utilized to indicate behaviors of the product and optimize the design for implementation, and security under various circumstances.

Importance of the CAD model

The first step in the FEA study is to the significance of the CAD model of the impeller analysis (Kashfi et al. 2022). A broad range of designs of CAD has been supported by ANSYS. That involves Pro or Engineer, CATIA, and Solid Works. Although, it is necessary to make sure that the model of CAD is clear and error-free otherwise it may cause problems during analysis.

Make a mesh of the impeller geometry

When the model of CAD is defect-free a mesh of discrete, small components has been developed via the ANSYS meshing mechanism (Pires et al. 2022). The mesh quality is important, for achieving precise results. Hence the mesh is distilled in places of increased pressure or deformation. Automatic meshing and manual meshing have been offered by the ANSYS, for generating an optimized mesh that grasps the physical and geometric manner of the impeller accurately.

Analysis of static structure

The analysis of static structure is used to predict how a material or structure will react to a variety of loading situations. In terms of an impeller, fixed structural analysis may get utilized to assess its stability and immobility under different loads (Zhang et al. 2023). The procedure includes developing a 3D model of the impeller, using the required limitation conditions and loading, and after that conducting the simulation to calculate stresses and strains. The results of this analysis may get used to optimize the impeller configuration and make sure that it will be able to resist the force-facing operations.

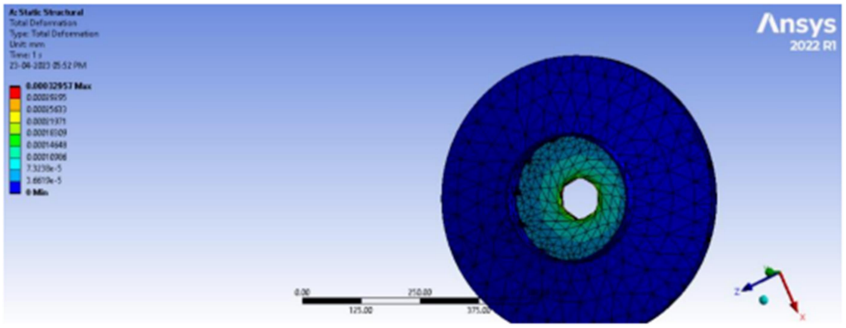

Fig 3: static structure

Analysis of modal

This is one type of structural analysis, which is used to detect one structure's mode of shapes and natural frequencies. A harmonic excitation, vibration result has been analyzed (Sadeghi et al. 2023). Modal analysis is generally used in structural design to make sure that the structure's natural frequency does not reach the frequency of excitation. Thus, this may induce resonance and the loss of structure.

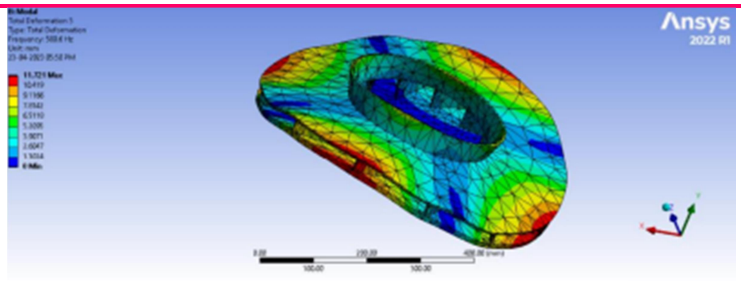

Fig 4: Total deformation

Quick thermal analysis

This is a type of analysis that falsifies the heat behavior of a system over a period of time. This incorporates the calculation of the allocation of the temperature of a procedure as it goes through transient thermal load like temperature change because of convection, transfer of the heat, and radiation.

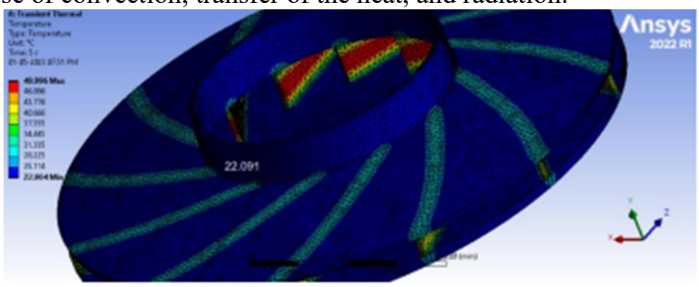

Fig 5: impeller's thermal profile

Computation Fluid Dynamic (CFD) analysis

1. Vista CPD analysis: a comprehensive suite of software tools has been offered by the Ansys, for the analysis and design of the pumps of centrifugal (Li and Zhang 2022). Engineers get the help of these tools to optimize the efficiency, performance, and durability, of the designs of the pumps, and also can decrease the cost and time of the development of the product.
2. BladeGen analysis: this is a tool of software that is used for optimizing and designing blades of turbomachinery (Cruz et al. 2022). Like, compressors, fans, and turbines, and also for maximizing the efficiency and performance level it gets optimized.

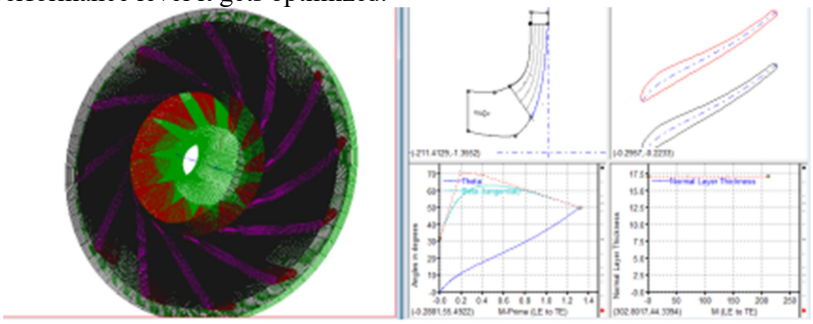

Fig 6: generation of blades

3. Volute analysis: in the design of turbomachinery, the Volute is a crucial part, which transforms the high-speed liquid leaving the impeller into a low-speed flow appropriate for downstream flow application (Gao et al. 2022). Ansys gives a complete suite of software tools that allow data transfer from the impeller design to the volute design seamless integration and the most satisfying performance.

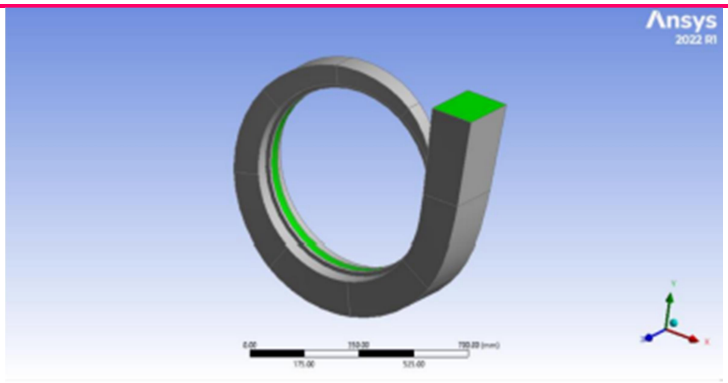

Fig 7: a 3D volute

4. Turbogrid analysis: turbo Grid is a software tool that is used to generate high-quality meshes for the application of turbomachinery (Qu et al. 2023). It has been designed to specifically develop meshes for radial, axial, and mixed-flow turbomachinery elements like vanes, blades, and impellers.

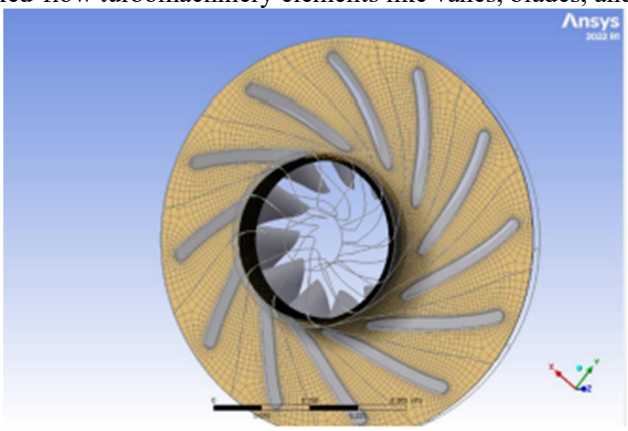

Fig 8: mesh generation of impeller

Conclusion and Recommendation

Based on the results that have been gained by the analysis of this study, in conclusion, it can be said that the blower impeller, volute, and casing design are appropriate for the planned application. The analysis of static structure shows that the casing and the impeller can resistance to time-generated forces and stress operation. On the other hand, the modal analysis results show that the volute and impeller have natural frequencies beyond the operating pace range. Additionally, the CFD analysis shows that the blower impeller can generate the expected results of pressure and airflow. Also, flow among the volute and casing is fairly consistent. The analysis of transient thermal shows that the temperature changes among the blowers in allowable limits, suggesting that the blower can work without being overheated continuously. This study's results may get used as a basis for future refinement and optimization of the ensign of the blower and may be used as a reference for further studies about blower design and analysis.

Future scope

Some of the possible future scopes of this study are:

1. Experimental validity: Although the results of this study acquired from the analytical tools of engineering are helpful, the experimental validation of the results is essential to make sure precision. The future work of this study may include the future performing experimentations to confirm the results that are acquired from the work. This may include, data investment, designing the setup, and statistical analysis.
2. Enhancement of performance: the future work may focus on increasing h performance of the blower beyond the necessities of the design. This can include modification optimizing impeller and volute aerodynamics, blade shape, and angle, and decreasing damage to the system.
3. Reduction of noise: during operation, blowers can cause notable noise. This may create a problem during specific applications. Future work may include analyzing the noise caused by the blower and

evolving techniques to decrease it. This use may involve sound-dampening materials, like barriers or acoustical foam.

4. Design optimization: the future scope of this study may involve optimizing the design of the impeller, volute, and casing founded on the results obtained from the analysis. This may get done by using many optimization techniques like genetic algorithms, response cover methods, and experimental designs.

References

1. Baskar, R.V. and Joseph, D.R., 2022, January. Design analysis of centrifugal compressor impeller vanes. In *AIP Conference Proceedings* (Vol. 2385, No. 1, p. 120006). AIP Publishing LLC.
2. Baskar, R.V. and Joseph, D.R., 2022, January. Design analysis of centrifugal compressor impeller vanes. In *AIP Conference Proceedings* (Vol. 2385, No. 1, p. 120006). AIP Publishing LLC.
3. Cruz, P.A.D., Yamat, E.J.E., Nuqui, J.P.E. and Soriano, A.N., 2022. Computational Fluid Dynamics (CFD) analysis of the heat transfer and fluid flow of copper (II) oxide-water nanofluid in a shell and tube heat exchanger. *Digital Chemical Engineering*, *3*, p.100014.
4. Gao, G., You, Q., Kou, Z., Zhang, X. and Gao, X., 2022. Simulation of the influence of wing angle blades on the performance of counter-rotating axial fan. *Applied Sciences*, *12*(4), p.1968.
5. Kashfi, M.E., Kouhikamali, R., Khayati, G. and Mahmoudimehr, J., 2022. CFD simulation of the laboratory-scale anaerobic digester to study the impacts of impeller geometric and operational parameters on its performance. *Available at SSRN 3957277*.
6. Li, H. and Zhang, Y., 2022. Lightweight design of the transonic centrifugal impeller based on fluid–structure coupling. *Physics of Fluids*, *34*(10), p.107109.
7. Li, H. and Zhang, Y., 2022. Lightweight design of the transonic centrifugal impeller based on fluid–structure coupling. *Physics of Fluids*, *34*(10), p.107109.
8. Odyjas, P., Więckowski, J., Pietrusiak, D. and Moczko, P., 2023. Challenges in the Design of a New Centrifugal Fan with Variable Impeller Geometry. *acta mechanica et automatica*, *17*(1), pp.16-27.
9. Pires, T., Dunlop, J.W., Fernandes, P.R. and Castro, A.P., 2022. Challenges in computational fluid dynamics applications for bone tissue engineering. *Proceedings of the Royal Society A*, *478*(2257), p.20210607.
10. Qu, W., Gong, W., Chen, C., Zhang, T. and He, Z., 2023. Optimization Design and Experimental Verification for the Mixed-Flow Fan of a Stratospheric Airship. *Aerospace*, *10*(2), p.107.
11. Ren, K.X., Shuai, Z.J., Wang, X., Jian, J., Yu, T., Dong, L.Y., Li, W.Y. and Jiang, C.X., 2022. Aerodynamic noise prediction of a high-speed centrifugal fan considering impeller-eccentric effect. *Engineering Applications of Computational Fluid Mechanics*, *16*(1), pp.780-803.
12. Sadeghi, M., Sontti, S.G., Zheng, E. and Zhang, X., 2023. Computational fluid dynamics (CFD) simulation of three–phase non–Newtonian slurry flows in industrial horizontal pipelines. *Chemical Engineering Science*, *270*, p.118513.
13. Spitzer, S., Grothe, R., Gude, M., Pohl, M. and Hermerath, P., Endless Fibre Reinforced Composite-Metal-Impeller: High Speed Burst Testing-Damage and Failure Analysis.
14. Stoyanov, V. and Nikolov, V., 2022, September. Numerical study of radial high-pressure blower with frequency converter. In *AIP Conference Proceedings* (Vol. 2449, No. 1, p. 050001). AIP Publishing LLC.
15. Zhang, F., Heng, Y., Liu, Q., Tao, R., Yang, W., Zhu, D. and Xiao, R., 2023. Optimization Design of a Semi-Open Centrifugal Impeller using GA-Fuzzy Logic Combination Strategy. *Journal of Applied Fluid Mechanics*, *16*(7), pp.1427-1441.