GK International Journal of Advanced Research in Engineering and Technology
E-ISSN:3048-8982 - Journal Website:https://www.gkijaret.com
Volume 1, Issue 1, August 2024, pp. 41-48

Enhancement of Air-Blower Performance by Modified Blades Configuration using Computational Fluid Dynamics (CFD)

R. Ganesamoorthy[1*]

[1]Professor, Department of Mechanical Engineering,
Chennai Institute of Technology, Chennai, 600069, Tamil Nadu, India
Email:ganesamoorthy72@gmail.com[*]

ABSTRACT: Air blowers are critical components in various industrial applications, including HVAC, pneumatics, and the aeration process, where efficient air movement is essential. This study focusses on evaluating the aerodynamic performance, including pressure distribution, velocity profiles, and flow patterns within the blower using atransient solver method. Three different turbine designs are modelled and investigated for their performance analysis. Spline and elevated spline turbines are proposed for this study's performance improvement, and these designs are compared to the narrow turbine designs. The results from ANSYS CFD provided detailed insights into the internal flow dynamics, enabling the identification of key parameters that influence the blower's performance, such as blade geometry and housing design. Through this analysis, valuable recommendations are offered for optimising air-blower designs to enhance their operational efficiency and reliability. The study reveals that the spline turbine design has a higher air mass flow rate compared to other designs. By utilising various impeller and housing geometries, further research is anticipated to aid in the creation of air blower designs that are more energy-efficient, which will have an impact on lower operating costs and increased sustainability in industrial operations. Our long-term objective is to reduce energy usage and maximise air blower performance. In the end, this will result in an economical and ecologically beneficial solution for a range of industrial applications.

Keywords: Impeller, Centrifugal Blower, Design of Blades, CFD, Performance of Air flow

DOI: https://doi.org/10.34293/gkijaret.v1i1.2024.5
Received 02 May 2024; **Accepted** 22 July 2024; **Published** 01 August 2024

Citation: R. Ganesamoorthy, "Enhancement of Air-Blower Performance by Modified Blades Configuration using Computational Fluid Dynamics (CFD)," *GK International Journal of Advanced Research in Engineering and Technology*, vol. 1, no. 1, pp. 41-51, Aug. 2024.

1. INTRODUCTION

Air-blowers are vital components in a wide range of industrial applications, including heating, ventilation, and air conditioning (HVAC) systems, pneumatic conveying, and various aeration processes. Their primary function is to move air or gas through systems, ensuring the necessary flow rates and pressure levels are maintained for efficient operation. The performance of an air-blower directly impacts the efficiency, energy consumption, and overall effectiveness of the systems in which they are employed. Centrifugal blowers are widely used in industries for ventilation and air circulation purposes. As per the discussion food industries primarily fruit juice producing company have a lot of issues while storing the fruits. Currently, cooling fans play an important role in thermal design and are designed in diversified specifications to meet various requirements for heat dissipation. Thus, the steady state of the food storage container is achieved with a centrifugal air blower. The following literature examines the problem statement and recent research development of air blower. According to the [1] the design calculation has been carried out and modelled using SOLIDWORKS. The primary objective of these projects is to enhance energy efficiency and ensure constant air flow delivery, enabling them to generate up to 84% efficiency. [2] analyses the performance of centrifugal fans using performance curves, revealing complex flow between blades and losses like entry, impeller, leakage, and volute losses. A model was created using Solid Work 2009, and performance analysis was carried out using experimental and ANSYS CFX software. The results were optimized using the Taguchi method.

[3] investigates the design of centrifugal blowers and aims to improve performance by using explicit methodologies. Parametric studies examine the impact of design parameters on pressure ratio and their interdependency. Experimental performance analysis shows better efficiency and flow rate in the current design compared to the industrial design. Centrifugal blowers are used to cool food before packing to avoid any spoilage occur. However, their continuous use at a low temperature can lead to corrosion and also affecting the performance of the centrifugal blower. Additional thus corrode state also affect the food and it became more dangerous to human health, [4] noticeable to improve the boilers conditions and conducted a study using CFD for fluid flow and FEA analysis.

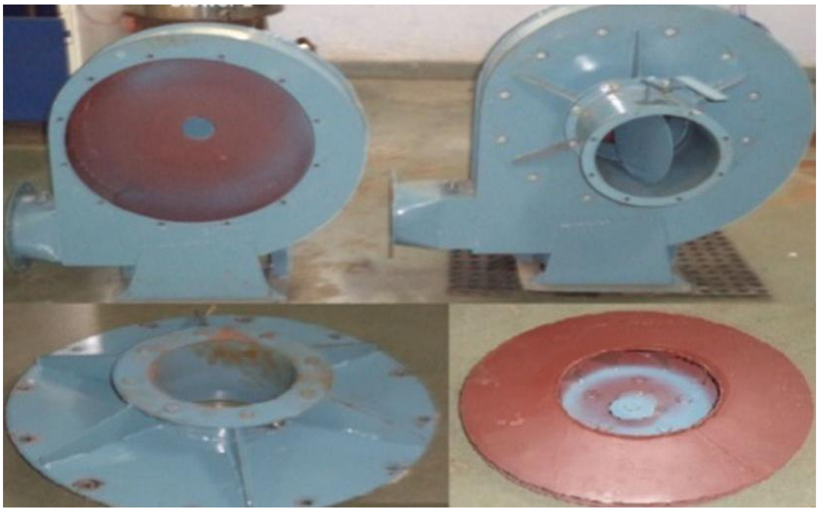


Figure 1: Pictorial view of manufactured blower [3]

Fruit juice producing companies face problems with their centrifugal blowers, which are made of M.S. material, which can cause corrosion and harmful fruit pulps. The weight of the blower makes it difficult to maintain food storage temperature. Using FEA and static and model analysis, [5] suggests optimising the centrifugal blower impeller for materials MS, SS, and SS304l. The outcomes are compared to the original design. According to the [6] studies the impact of geometric parameters on centrifugal fans with backward and forward-curved blades using an experimental setup, CFD, and prototype model, examining parametric studies like quantity power, flow, efficiency, and flow coefficient. According to the [7] the performance of a centrifugal blower is improved through optimization using the Taguchi method and ANOVA approach. Numerical simulations and matrix experiments show improved performance, with a 7.4% higher efficiency compared to the original configuration.

[8] investigate the impact of an enlarged impeller on the performance of a G4-73 type centrifugal fan. Comparing the original impeller diameter with the two larger impellers with varying outlet diameters. the result shows that the larger impeller has more volute loss, flow rate increases, pressure rises, and an increase in noise level also decreases efficiency. A reduced impeller-volute gap is the cause of greater noise levels, according to the noise frequency study. [9] develops high performance centrifugal fans to increase the system performance and space utilization of computer devices. The performance of the system can be evaluated under different operating conditions. The numerical results are utilized for comprehensive flow visualization, torque

calculation, efficiency estimation, and noise analysis, with the fan performance curve and sound pressure level spectrum aligning with the simulations. The study suggests two modification options based on flow visualization at each operating point to improve fan performance through numerical calculation. [10] looked at adding distinct chambers upstream of the impeller to improve the centrifugal compressor's working stability. A shear stress transport turbulence model and three-dimensional Navier-Stokes equations were solved using ANSYS-CFX 15.0 for aerodynamic study. Three geometric features were chosen as design elements and the compressor failure functioned as the objective function. Radial basis neural network model was constructed as a stand-in model and the Latin hypercube sampling approach yielded 27 design points for the design space. The centrifugal compressor with the best discrete cavities improved its stall margin by 15.4% according to the results.

[11] assessed the velocity measurement and the numerical flow simulation findings for a single stage centrifugal fan. Two-dimensional instantaneous velocity is measured using particle image velocimetry (PIV) and stable and unsteady computations are performed using the commercial tool CFX-TascFlow. The results demonstrate that the flow regime around the diffuser hub includes the volute. The flow behavior of an impeller diffuser is predicted by [12] using a numerical technique. The findings indicate that while diffusers improve energy transformation, impeller perform better at converting energy. Greater gaps improve efficiency and offer a thorough knowledge of the relationship between the impeller and diffuser as well as fan performance.

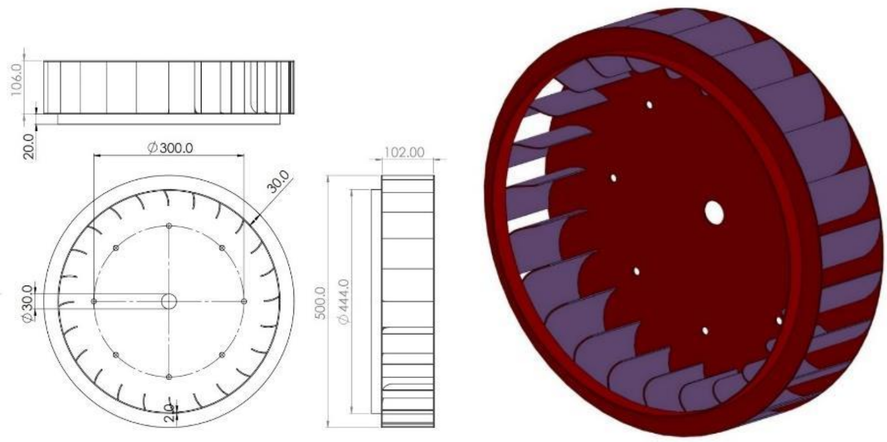


Figure 2: Impeller drafting and 3d design [14]

According to the influence of geometric factors [13] examine the centrifugal pump volute’s performance. By using a numerical approach, the volumetric area, cross-section form, spiral development area and radial gap between the volute tongue and impeller are all evaluated. The results indicate that a geometrical parameter is a major factor in determining a pump’s efficiency. [14] has attempted to explain the fundamental principles of centrifugal air blower design as well as the experimental configuration and manufacturing process. Technical specifications and variations in the air blower's design and apparatus are also presented in the project. The project's goal is to offer a thorough comprehension of the air blower's functionality. [15] did research on two distinct volute case designs for a centrifugal blower with an impeller shrouded in reverse blades. At the entrance, the flow fields were examined at different points within the volutes while the throttle was fully open. The flow characteristics were examined using a three-dimensional probe, and the pressure recovery coefficient and loss coefficient findings were reported. [16] used ANOVA and the Taguchi method to optimize the blower volute and the performance of a centrifugal blower. Geometrical parameters are prioritized using the three-level analytic hierarchy technique. Matrix experiments employ numerical simulations with FLUENT and Minitab software. Compared to the current blower, the improved arrangement performs better, with a 7.4% greater efficiency at the design rated speed.

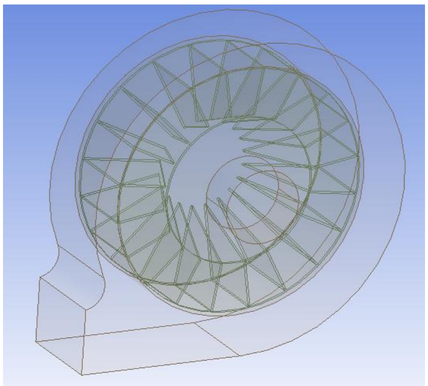
[17] compared numerical simulations of backward-curved airfoil centrifugal blowers with experimental data. A 15.1% efficiency deviance and 4.8% static pressure were displayed in the results. An improved design

with a 1.5% efficiency gain and 7.9% static pressure was the result of proposed parameter modifications. [18] used a two-component frequency-shifted laser velocimeter to measure the turbulence and velocity profiles of a four-vaned radial flow impeller in a volute pump. Turbulence intensities varied from 0.06 to 0.28, with the unshrouded impeller and low flow rates exhibiting the greatest values, according to the data. There was greater isotropic turbulence at higher flow rates. [19] presents detailed measurements of swirling flow in a centrifugal compressor volute, revealing variations in velocity, pressure distribution, and flow structure.

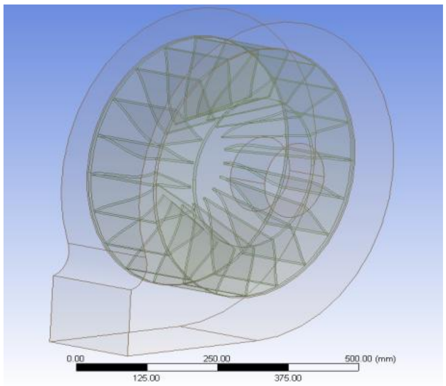
The literature given above summarizes the various tests conducted on the blower to study the efficiency of the blower on various parameters such as blade shape and geometry change, material change, impact flow rate analysis and pressure distribution of centrifugal air-blower. All these tests are carried out to improve the performance of the blower. The main objective of this study is to improve the performance of the air flow by changing the physical Geometry parameter.

2. DESIGN METHODOLOGY

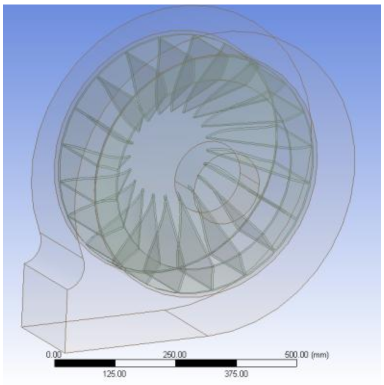
Probably the most important contribution of the proposed work is towards improving the performance of the blower and also improving the air flow with changes in geometric conditions. Figure 3 shows the design of air blower models 1 (narrow turbine), 2 (spline turbine), and 3 (elevated spline turbine). Figure 4 shows the mesh geometry of the proposed models, and the tetrahedrane mesh method is used in this work. Boundary conditions are illustrated in Figure 5. The simulations were conducted using Ansys Fluent software to analyse the flow behaviour and performance of each blower model. The impeller rotational speed (2850 RPM) and other boundary conditions (wall functions) are set to replicate real-world operating conditions. time dependent transient k-epsilon solution model was used for solver processing and Coupled solution method was employed in this work.



a) Model 1



b) Model 2



c) Model 3

Figure 3: Proposed design of blower

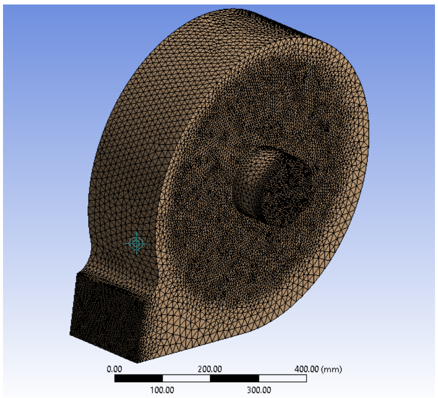


Figure 4: Mesh geometry of proposed models on CFD fluent

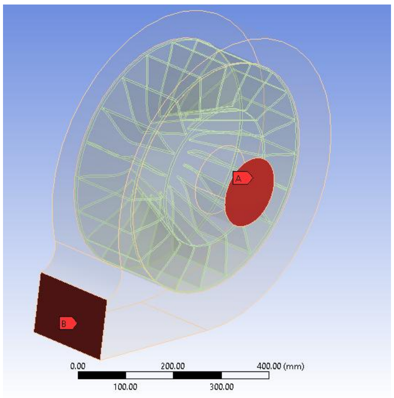


Figure 5: Boundary conditions A (inlet) & B (outlet)

3. RESULTS AND DISCUSSIONS

Simulation results are evaluated for all proposed designs, and contour results are presented in the below figures. Due to the fluid motion condition, the dynamic pressure and total pressure are considered for the evaluation process. A rotational motion has been generated among fluids due to impeller motion. To validate the fluid flow conditions, the turbulent kinetic energy (TKN), velocity magnitude, and axial velocity results are taken into account.

3.1 MODEL 1 (NARROW TURBINE)

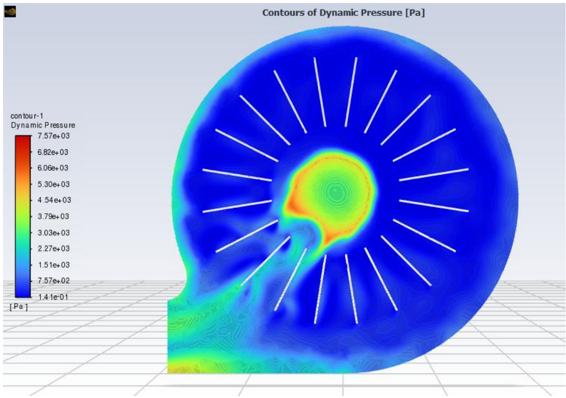


Figure 6: Contour results of dynamic pressure (Model 1)

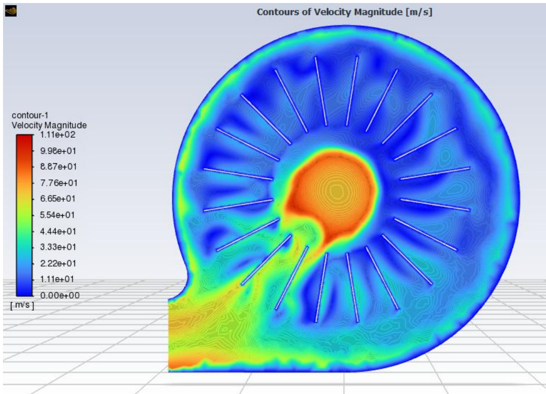


Figure 7: Contour results of velocity magnitude (Model 1)

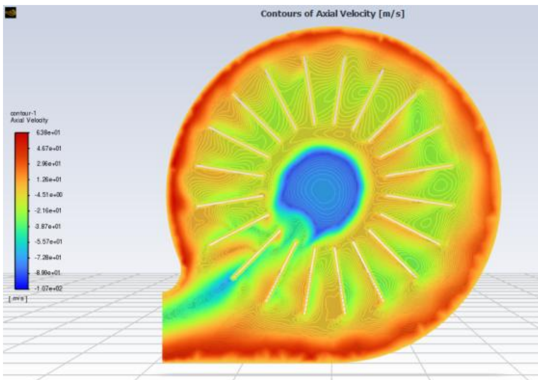


Figure 8: Contour results of axial velocity (Model 1)

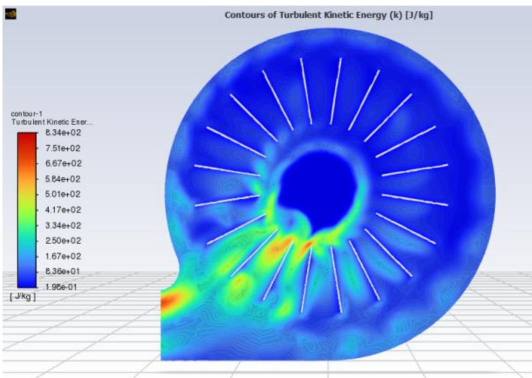


Figure 9: Contour results of turbulent kinetic energy (Model 1)

3.2 MODEL 2 (SPLINE TURBINE)

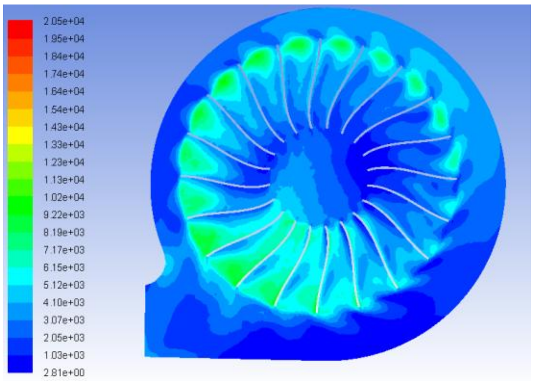


Figure 10: Contour results of dynamic pressure (Model 2)

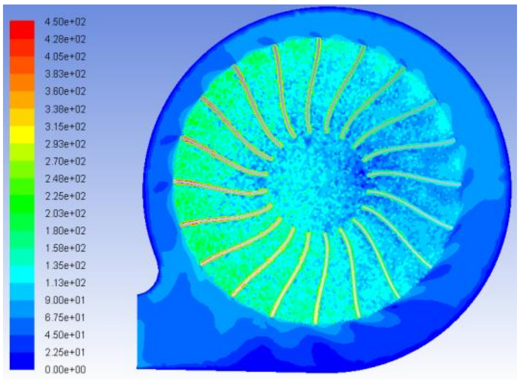


Figure 11: Contour results of velocity magnitude (Model 2)

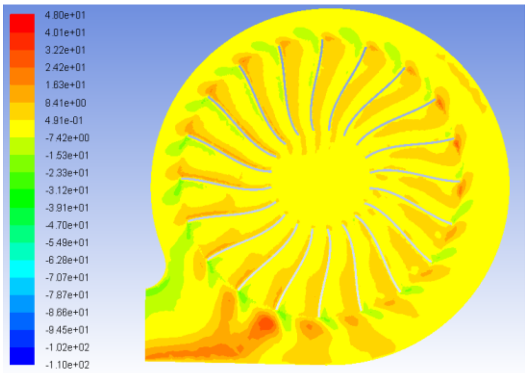


Figure 12: Contour results of axial velocity (Model 2)

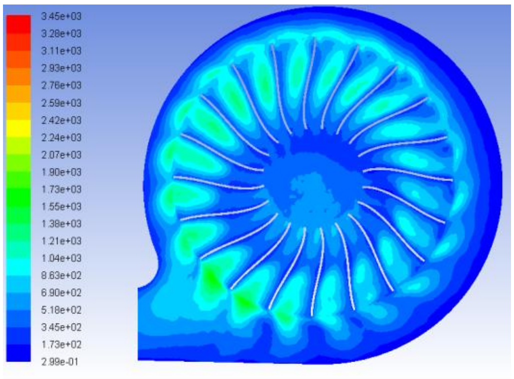


Figure 13: Contour results of turbulent kinetic energy (Model 2)

3.3 Model 3 (Elevated Spline Turbine)

The axial velocity, turbulent kinetic energy, dynamic pressure, and velocity magnitude contour data for the narrow turbine model are shown in Figures 6 through 9. Figures 10 to 13 demonstrate the axial velocity, turbulent kinetic energy dynamic pressure and velocity magnitude contour findings for the spline turbine model.Figures 14-17 show the contour findings for turbulent kinetic energy, axial velocityand dynamic pressure for the narrow spline turbine model in that order. The contour results demonstrate how the flow characteristics of several turbine types vary, pointing to changes in efficiency and performance. These numbers offer important information for maximizing each turbine model's design and functionality in order to improve performance as a whole. The flow characteristics for each turbine types are summed up in Table 1 below, facilitating direct comparisons between the various variants. The table provides a thorough summary of the simulation findings, emphasizing significant variations in performance indicators such turbulent kinetic energy and velocity output. Researchers may decide how to enhance the operation and design of each turbine type for maximum performance by examining the contour plots as well as condensed data in Table 1.

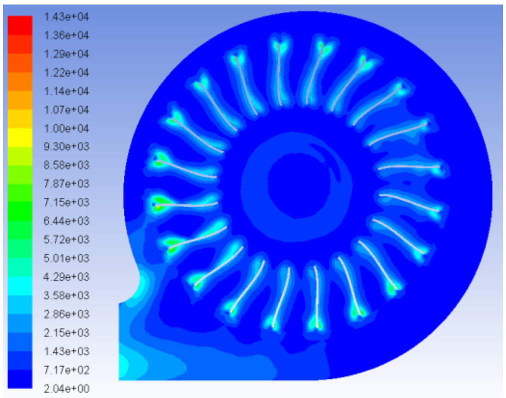


Figure 14: Contour results of dynamic pressure (Model 3)

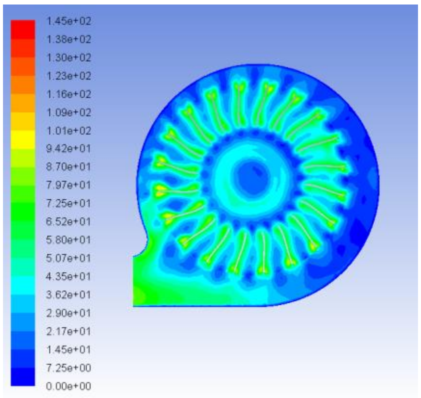


Figure 15: Contour results of velocity magnitude (Model 3)

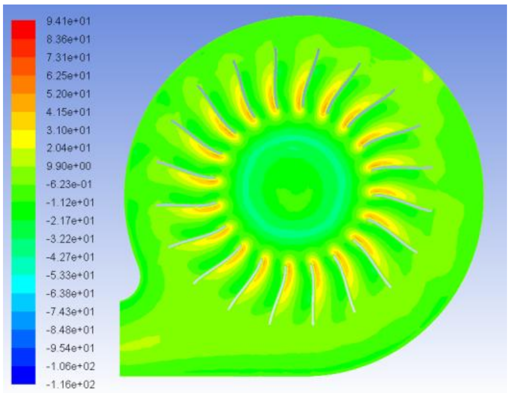


Figure 16: Contour results of axial velocity (Model 3)

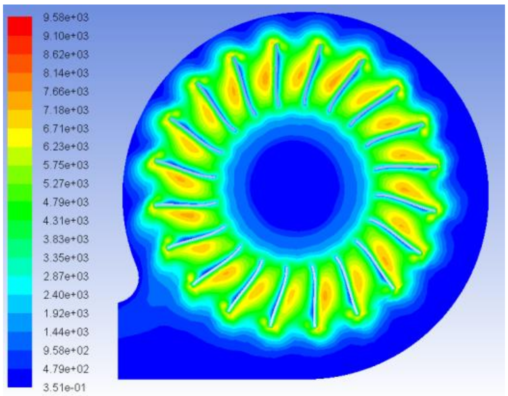


Figure 17: Contour results of turbulent kinetic energy (Model 3)

Table 1: Comparison of different impeller designs

	Units	Narrow turbine	Spline turbine	Elevated spline turbine
Dynamic pressure	Pa	3299.5888	3242.6944	2930.969
Total pressure	Pa	3371.2328	3288.7176	2962.2328
Velocity	m/s	71.0000	71.8116	68.864
Volume Flow Rate	m^3/s	1.672891	1.676894	1.6756516
Turbulent kinetic energy	j/kg	308.1479	453.3743	302.8963

4. CONCLUSION

The outcomes of this study have been validated that CFD is a practical method for optimizing air blower designs and analyses. Performance bottlenecks which can occasionally be difficult to locate using more traditional approaches can be more easily identified and addressed because to CFD's ability to analyze complex fluid dynamics in great detail. The results also highlight the need of selecting boundary conditions and turbulence models with care to ensure accurate simulation outcomes. Taken as a whole, this work enhances the ongoing efforts to design more energy and cost-efficient air blower systems. A route towards improved industrial sustainability can be found by applying the approaches and knowledge gained from this study to a variety of blower designs and operating situations. Subsequent research endeavors may concentrate on verifying the CFD outcomes through experiments and investigating cutting-edge materials and production methods to enhance blower efficiency. In comparison to other models, the Spline Turbine (Model 2) performs better based on the results of the CFD simulation. The results of the simulations provide valuable insights into the effectiveness of the proposed design modifications on blower performance. These insights can help engineers make informed decisions to optimize blower efficiency and overall performance.

REFERENCES

[1] S. R. Kshirsagar, M. Shinde, S. Nandgude, D. Salunkhe, and P. Magar, "Design Analysis and Optimization of Centrifugal Blower," *International Journal of Innovations in Engineering Research and Technology*, 2020.

[2] K. Patel, and P. M. Patel, "Performance Analysis and Optimization of Centrifugal Fan," *International Journal of Emerging Trends in Engineering and Development*, vol. 2, no. 3, 2013, pp. 261-270.

[3] B. D. Baloni, S. A. Channiwala, and S. N. R. Harsha, "Design, Development and Analysis of Centrifugal Blower," *J. Inst. Eng. India Ser. C*, vol. 99, 2018, pp. 277-284.

[4] A. Salunkhe, P. Pawar, S. Ghodke, and S. Pawar, "Analysis and Optimization of Centrifugal Blower Material by using FEA & CFD," *International Research Journal of Engineering and Technology*, vol. 8, no. 6, 2021.

[5] S. V. Kesare, and M. C. Swami, "Analysis and optimization of Centrifugal Blower by using FEA," *International Journal of Engineering Development and Research*, vol. 4, no. 3, 2016.

[6] O. P. Singh, R. Khilwani, T. Sreenivasulu, and M. Kannan, "Parametric Study of Centrifugal Fan Performance: Experiments and Numerical Simulation," *International Journal of Advances in Engineering & Technology* vol. 1, no. 2, pp. 33-50, 2011.

[7] B. D. Baloni, Y. Patak, and S. A. Channiwala, "Centrifugal Blower Volute Optimization Based on Taguchi Method," *Computers & Fluids*, vol. 112, pp. 72-78, 2015.

[8] L. Chunxi, W. S. Ling, and J. Yakui, "The performance of a centrifugal fan with enlarged impeller," *Energy Conversion and Management*, vol. 52, no. 8-9, pp. 2902-2910, 2011.

[9] S.-C. Lin, and M.-L. Tsai, "An integrated performance analysis for a backward-inclined centrifugal fan," *Computers & Fluids*, vol. 56, pp. 24-38, 2012.

[10] S.-B. Ma, and K.-Y. Kim, "Optimization of Discrete Cavities in a Centrifugal Compressor to enhance Operating Stability," *Aerosp. Sci. Technol.*, vol. 68, pp. 308-319, 2017.

[11] T. Meakhail, and S. O. Park, "A Study of Impeller-Diffuser Volute Interaction in a Centrifugal Fan," *Journal of Turbomachinery*, vol. 127, no. 1, pp. 84-90, 2005.

[12] K. V. Karanth, and N. Y. Sharma, "CFD Analysis on the Effect of Radial Gap on Impeller-Diffuser Flow Interaction as well as on the Flow Characteristics of a Centrifugal Fan," *International Journal of Rotating Machinery*, 2009.

[13] S. Yang, F. Kong, and B. Chen, "Research on Pump Volute Design Method Using CFD," *International Journal of Rotating Machinery*, 2011.

[14] M. A. Kattimani, M. N. Ahmed, and A. Akram, "Design and Fabrication of Centrifugal Air Blower Test Rig," *International Journal of Thermal Engineering*, vol. 10, no. 1, pp. 11-18, 2022.

[15] B. D. Baloni, S. A. Channiwala, and V. K. Mayavanshi, "Pressure recovery and loss coefficient variations in the two different centrifugal blower volute designs," *Applied Energy*, vol. 90, no. 1, pp. 335-343, 2012.

[16] B. D. Baloni, S. A. Channiwala, and V. K. Mayavanshi, "Centrifugal Blower Volute Optimization based on Taguchi Method," *Computers & Fluids*, vol. 112, pp. 72-78, 2015.

[17] C.-K. Huang, and M.-E. Hsieh, "Performance Analysis and Optimized Design of Backward-Curved Airfoil Centrifugal Blowers," *HVAC&R Research*, vol. 15, no. 3, 2009.

[18] R. D. Flack, C. P. Hamkins, and D. R. Brady, "Laser velocimeter turbulence, measurements in shrouded and unshrouded radial flow pump impellers," *International Journal of Heat and Fluid Flow*, vol. 8, no. 1, pp. 16-25, 1987.

[19] E. Ayder, R. V. den Braembussche, and J. J. Brasz, "Experimental and Theoretical Analysis of the Flow in a Centrifugal Compressor Volute," *Journal of Turbomachinery*, vol. 115, no. 3, 1993.