International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

Design, Optimization and Relocation Strategy for GE Gas Turbine Exhaust Frame Blowers to Reduce Overheating and Operational Trips.

Hassan Lawal Bagudo, Ashiedu Festus Ifeanyi, Emagbetere Eyere
Department of Mechanical Engineering,
Federal University of Petroleum Resources Effurun, Nigeria.

Abstract - This research study aims at solving industrial turbine issues using a relocation strategy by evaluating the existing blower at Warri refining and petrochemical company and comparing with a modified blower, designed and simulated to provide a permanent solution to the costly way of repairing and replacing the electric motors and blowers bearing of the 88TK-O1/02 in Warri Refining and Petrochemical Company and in extension to the entire sub Saharan African leading to cost effectiveness and enhance safety in the industry. The research method involves collection of data from the blower design specification at WRPC with the focus on the speed, frame blade diameter, designed operating temperature and frame mesh. This was to ascertain the behavior of the existing blower and establish the extent of the challenges in the entire turbine system. After which, a new blower model was designed using solid works with some specified dimensions of an impeller with 9600mm diameter, inlet blade of 35° and an outlet angle of 26°with two different modification setups. The result indicated that in comparison to the original set up, the second modification achieves the same output parameters with a higher angular velocity requirement of about 3600 RPM, this indicates that the modified blower design optimizes colling efficiency compared to the existing blower. The CFD analysis shows that the backward inclined impeller geometry presents a stable, high-volume airflow that effectively reduces the risk of overheating under the challenging ambient weather condition exposure in the Niger Delta region. This study solves an impending long-term practical issue on the gas turbine plants at WRPC and extends its application to countries with high climate temperature 30° and above. In the future, this study might be a tender manual to a new trend of Gas turbine purchase agreement. It will be a reason for buyers to demand turbines from original equipment manufacturers (OEM) along with this modification.

Keywords: Gas turbine, Blower, Thermal management, CFD simulation, WRPC, Reliability, Gas turbine cooling.

1 INTRODUCTION

Gas turbine is the central component of advanced power generation due to its high demand for fast response load differences in industrial applications, for both electric and thermal energy production. The Gas turbine operate by sucking in atmospheric air then pass it through 17 stages compressor and the compressed air moves to a combustion chamber where ignition & fuel are introduces to produces higher pressure and higher velocity gas(flue gas) after combustion, this gas moves with its high temperature & pressure properties finding its way out through the turbine buckets(blades) and by so doing turning the shaft of the turbine thus producing a rotary motion that turns an Electric generator. In as much as the successful achievement of enhancing the overall efficiency of the gas turbine cycle is fundamentally based upon the performance efficiency of each individual component, along with the synergistic interactions that occur among all the components functioning collectively as an integrated engine system (Lakshya Kumar, 2019), the exhaust frame blower happens to be one of the major components in ensuring the operational efficiency and reliability of the Gas turbine. These blowers aid removal of hot exhaust gases and maintain smooth ventilation within the Gas turbine exhaust frame, preventing overheating and thermal stress on the components surrounding the entire gas turbine. However, despite the importance of the exhaust frame blower in ensuring efficiency of turbine operations, the persistent issues of overheating and tripping continue to afflict the gas turbine located in areas with high climate temperature such as the Warri Refining and Petrochemical Company, thereby compromising overall operational integrity. These disruptive events frequently occurs as a result of insufficient airflow volume or inadequate velocity generated by the blower when subjected to varying turbine loads, combined with poorly directed airflow within the exhaust frame that fosters the development of localized hot spots, alongside inefficient heat dissipation that comes from the inherent mechanical limitations associated with the current blower design, as well as delays or outright failures in the thermal trip logic integrated within the turbine's control system.

Exhaust frame blowers, which are mechanical devices designed to move gases such as air by harnessing the kinetic energy imparted by a rotating impeller, are made up of two important components: housing, which encases the mechanism, and the rotor, which is the component responsible for the fluid movement. While it is common for these blowers to be powered by electric motors, it is important to note that alternative energy sources can also be utilized to operate them effectively. The application of exhaust frame blowers across several industrial sectors demonstrates their versatility and importance in various processes. During the operational phase, the fluid is collected through either a volute, a spiral-shaped passage designed for efficient flow or a series of diffusing passages, subsequently exiting the impeller at an elevated velocity and pressure, which is critical for many applications. The effect of these issues is a detrimental cycle characterized by frequent periods of downtime, emergency shutdowns that interrupt

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

productivity, the costly replacement of essential components, and a consequential reduction in turbine efficiency that covers the overall operational performance.

This study presents a novel solution to overheating and trips for gas turbine exhaust frame blower by incorporating a relocation strategy to the existing blower with the design taking into consideration both real and practical situations. The study covers a model that was designed and simulated with the blower relocated outside the turbine hall as against the initial position where the blower operates above combustion chamber which is usually very hot. Operating the blower at a high temperature burns out the motor and bearings of the blower causing a significant challenge in power generation systems, affecting performance and component longevity. Hence, advanced cooling techniques are considered for maintaining optimal turbine rotor inlet temperatures, which can exceed 1700°C in modern systems (Han, 2018). The purpose of this paper is to analyze the performance of the existing frame blower; and compare the simulated results of the relocated exhaust frame blower with the existing blower.

Researchers increasingly focused on improving blower performance through design optimization, enhancing thermal management within the existing configuration, material and structural improvements. This study introduces a fundamentally different approach which changes the operating environment of the blower from a high temperature recirculating airflow zone to a cooler external zone outside the turbine hall rather than modifying the blower. Over the years, researchers have investigated multiple techniques to enhance the efficiency of the exhaust frame blower of a gas turbine. (Azzawi, 2023) discusses the design optimization of a subsonic blower wind tunnel by employing a large-scale CFD model to analyze the entire system rather than just the test section. It calculates losses from each component to determine the power requirements for operation. The study emphasizes the importance of flow conditioners, such as honeycomb and mesh screens, in enhancing flow uniformity and reducing turbulent intensity, which are critical factors in optimizing blower design for aerodynamic research. The paper does not explicitly mention any limitations regarding the design and characterization of the blower wind tunnel, focusing instead on the design process and validation of results through experimental and numerical simulations. While the study emphasizes the impact of flow conditioners on flow quality, it does not address potential limitations related to the scalability of the wind tunnel design or the applicability of the results to different aerodynamic research scenarios. Furthermore, (Cai, 2019), focuses on enhancing self-physical cooling through an innovative structure. Key features include a motor connected to a blowing device, cooling fins on the device's outer wall, and a cooling device positioned around the fins. The drainage cover facilitates efficient air drainage, allowing air to swiftly pass over the cooling device for effective heat dissipation. This design ensures improved cooling performance for the air blowing device, ultimately enhancing the blower's efficiency and longevity. In a parallel effort, (Kreidler et al., 2017) discusses a blower design that includes impeller housing, an impeller, and an electric motor with a unique motor housing structure. The design features a venturi chamber that creates a pressure difference between the impeller and motor housings, facilitating airflow to cool heat-generating electrical components. This optimization enhances the blower's efficiency and performance by ensuring effective cooling, which is crucial for maintaining operational reliability and extending the lifespan of the electrical components within the motor. (Nasrollahi & Sabbagh Yazdi, 2024) emphasizes optimizing blower design through the integration of various airflow modification techniques in a multi-fan blower type wind tunnel. Key components such as bell-mouth intakes, propeller caps, and honeycombs were added to enhance airflow velocity and reduce turbulence. Tests at different engine speeds demonstrated significant improvements in flow uniformity and reductions in turbulence intensity, achieving up to 42% reduction in turbulence with minimal airflow velocity loss, thus optimizing the wind tunnel's performance for civil engineering applications. The study mentions a reduction in airflow velocity as a limitation when implementing various equipment to improve flow uniformity in the wind tunnel. For instance, the improvements achieved in reducing turbulence intensity came at the cost of a 21%, 23%, and 12% reduction in airflow velocity at propeller rotation speeds of 600, 1000, and 1200 RPM, respectively. Another limitation highlighted in the paper is the trade-off between improving flow uniformity and maintaining airflow velocity distribution parameters within the downstream section of the flow. While significant improvements of approximately 55% and 25% were achieved in uniform air velocity distribution parameters, this enhancement was balanced against the reduction in airflow velocity caused by the modifications. In a different direction (Jang & Jeon, 2014) focuses on two key variables in their study: the bending angle of the impeller and the blade thickness of the impeller tip. By adjusting these parameters, the performance of a regenerative blower used for a 20-kW fuel cell system is enhanced, achieving a pressure increase of up to 2.8% and an efficiency improvement of up to 2.98% compared to the reference blower. The optimal configuration is identified at an impeller thickness of 2 mm and a bending angle of 14 degrees. The study primarily focuses on two design variables, the bending angle of the impeller and the blade thickness of the impeller tip, which may limit the exploration of other potential design parameters that could further enhance the performance of the regenerative blower. The numerical simulation results, while validated against experimental data, indicate a maximum error of 5 percent, suggesting that there may be discrepancies between the simulated and actual performance, which could affect the reliability of the optimization results. (Hoffman & Lathrop, 2007) worked on a blower assembly design optimized through the integration of a stator assembly within the blower housing, utilizing a brushless D.C. motor configuration. The impeller features vans with optimized heights to reduce fluid eddy losses during air compression. Additionally, a precisely sized bearing assembly maintains a minimal air gap of approximately 0.0002 inches between the vanes and the housing interior surface, enhancing efficiency and performance during operation. This design approach significantly contributes to overall blower efficiency. Similarly to the aforementioned, (Peng, 2022) discusses the design optimization of a subsonic blower wind tunnel by employing a large-scale CFD model to analyze the entire wind tunnel system. It calculates losses from each component to determine the power requirements for operation. The study emphasizes the importance of boundary layer controllers, such as honeycomb and mesh screens, in enhancing flow uniformity and reducing turbulent intensity, which are critical factors in optimizing blower design for improved aerodynamic research outcomes. The paper does not explicitly mention any limitations regarding the design and characterization of the blower wind tunnel, focusing instead on the design process and validation of results through experimental measurements. While the study highlights the impact of boundary layer controllers on flow quality, it does not address potential limitations related to the scalability of the CFD model or the accuracy of the experimental validation methods used.(Zhang et al., 2024) research study highlights that gap circulating flow significantly impacts blower performance, suggesting that optimizing the gap size can enhance efficiency. Under design conditions, efficiency decreases by 5.3-8.2% due to power loss maintaining circulating flow, despite a 12% increase in impeller efficiency calculated by net flow rate. Therefore, optimizing the gap can balance the benefits of increased flow rate against efficiency losses, ultimately leading to improved blower design and performance under various operating conditions. The efficiency of the blower is decreased due to the circulating flow, which consumes a portion of the power needed for blower operation. Under design conditions, this efficiency reduction ranges from 5.3% to 8.2%, depending on the sizes of the gaps between the impeller and volute casing. Although the gap circulating flow increases the net flow rate of the impeller by about 12% compared to the inlet flow rate, this increase in flow rate does not compensate for the overall efficiency loss, indicating a trade-off between flow rate and efficiency in blower performance. Also, (Herman, 2018) discusses a downhole-blower system designed to enhance production in a wellbore. While it does not provide specific details on blower design optimization, it emphasizes the integration of a blower with an electric machine to improve efficiency and output. The focus is on the operational synergy between these components to maximize production, suggesting that design considerations should prioritize compatibility and performance in the wellbore environment for optimal results. While (Hara & Nishikawa, 2020) focuses on enhancing the efficiency of the electric motor and the impeller configuration. Key aspects include the labyrinthine air flow channel formed between the flange and the impeller, which not only facilitates effective cooling of the armature but also traps abrasion powder from the commutator and brush. This design minimizes wear and improves performance, ensuring a consistent air current generation while maintaining the longevity of the components involved in the blower system. (Rini & Saarloos, 2015) went forward to work on blower design optimization which involves establishing geometric variables and utilizing aerodynamic similarity to enhance efficiency. Computational Fluid Dynamics (CFD) iterations were employed to minimize losses and improve performance by adjusting the blower's geometry. The design achieved a pressure rise of 5 in H_2O while consuming less than 15 W of power. The final design met weight and volume constraints, and a prototype is ready for testing to validate CFD outputs and further refine the design. The design of the Personal Air Ventilation System (PAVS) blower is constrained by size and weight requirements, specifically needing to fit within a volume of less than 60 in^3 and weigh less than 2 lbs, which may limit the complexity and performance of the blower design. The analysis and performance estimates of the blower rely on aerodynamic similarity and computational fluid dynamics (CFD) simulations, which may not account for all real-world variables and conditions, potentially affecting the accuracy of the efficiency and performance predictions. (Lichtblau, 1988) discusses a single-stage axial blower design that optimizes airflow delivery by utilizing a combination of a fixed stationary guide wheel and impellers with varying hub diameters, while maintaining the same outer diameter. This structural combination allows for a series of blowers to produce a wide range of delivery flow rates, making them suitable for different air-cooled internal combustion engines with varying cylinder numbers and cooling requirements, thus achieving effective blower design optimization. (Song et al., 2005) worked on blower design optimization which involves a centrifugal fan, a driving unit, and a bracket with a protrusion that enhances airflow. The fan housing features a suction port and a discharge port, with the latter designed in a 'V' shape to improve air discharge. The inclusion of curved inclination surfaces on the discharge port increases the discharge area, allowing for a more uniform flow rate, which reduces noise and enhances the volume of air discharged, optimizing overall performance. (Thomas et al., 2019) focuses on optimizing the control of soot blowing in power plants using fuzzy logic techniques rather than the design of the blower itself. It highlights that traditional soot blowing operates on a fixed schedule, which can lead to unnecessary blowing and efficiency loss. By optimizing the fuzzy controller parameters, the research aims to enhance the efficiency of the soot blowing process, ensuring that only necessary boiler stages are blown, thus improving overall power plant performance. And finally, (Bagudo et al., 2025) worked on solving industrial turbine issues by relocating the blower system to a cooler area and validate its performance using computational fluid dynamics simulations to model the airflow. While this study has some similarities with this research there was no simulation comparison between the existing blower system and the modified one to validate the result. However, the simulation results provided a detailed analysis of the blower's performance, showing its potential to deliver the required volumetric flow rate and static pressure necessary to effectively cause heat from the gas turbine exhaust frame.

Although blower optimization in Gas turbine operations has been widely studied, this addresses the gap by presenting a novel approach in the relocation of the blower. This study possesses better functionality than the aforementioned. Unlike previous study that focused on optimizing the blower design, airflow part and internal cooling workings within the enclosed turbine, this work uniquely relocates the exhaust frame blower to a cooler external zone. This relocation strategy alters the thermal boundary conditions experienced in the existing blower, thereby reducing overheating and Gas turbine trips. In this study, the solution is validated through CFD simulations comparing the existing and modified configurations, demonstrating optimized cooling efficiency and reliability.

2. WORKING PRINCIPLES OF GAS TURBINES

Gas turbines operate by sucking in atmospheric air then pass it through 17 stages compressor and the compressed air moves to a combustion chamber where ignition & fuel are introduces to produces higher pressure and higher velocity gas(flue gas) after combustion, this gas moves with its high temperature & pressure properties finding its way out through the turbine buckets(blades) and by so doing turning the shaft of the turbine thus producing a rotary motion that turns an Electric generator of 32MW capacity in this case.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

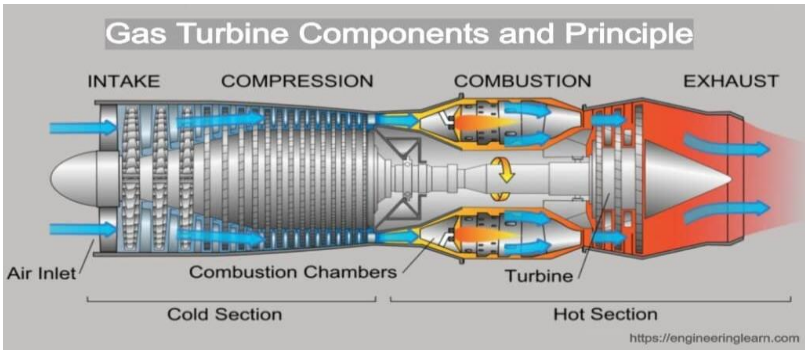

Figure 1: Gas turbine components and principles

The gas turbine machine will be impossible without the cooling and sealing air system because higher pressure air coming from another stage of compressor is used for this purpose. The exhaust frame blower air requires much lower air pressure to necessitate the use of another air supply equipment (air blower). The need for air cooling to the exhaust material is a must due to the highly exhausted temperature (averaging 300°C to 600°C). This is where the exhaust frame blowers tag as (88TK- 01/02) comes in with the aim of drawing in ambient air and forcing it into the frame cavity of the exhaust to cool the area surrounding the rear bearing and other components sensitive to heat. This cooling mechanism is important for maintaining the reliability of the turbine because the exhaust frame sits directly in the path of the hot exhaust gases exiting the turbine.

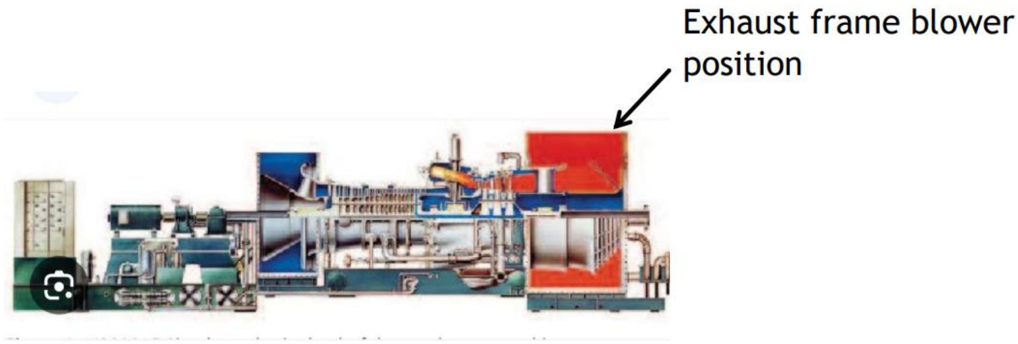

Figure 2: GE Gas turbine MS 5001

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

Figure 3: GE Power plant

Figure 4: Exhaust frame blower

3. SIMULATION SETUP

The Blower model was designed using SolidWorks using the specified dimensions highlighted in Table 1. The impeller was also designed using the specified impeller diameter of 9600mm, backward inclined, with blade inlet and outlet angles of 35 and 26 degrees respectively. The designed impeller is shown in figure 5 below while the designed blower casing is shown below in figure 6

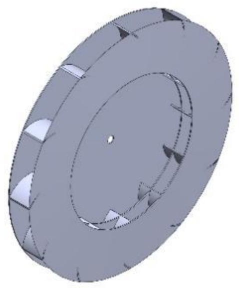

Figure 5: Impeller

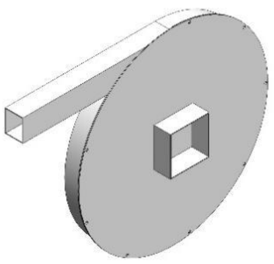

Figure 6: Blower Casing

A fluid region with diameter slightly larger than that of the casing, designed to envelope the casing and serve as the region of fluid flow around the impeller. To account for the piping system for the fluid horizontal and vertical travel, horizontal and vertical pipes of 12,330mm and 10,895mm respectively with a pipe cross sectional dimension of 110 by 140mm.

Table 1: Highlight of blower design specifications

Materials Used	**Dimensions**
Blade frame diameter	120mm
Wheel blade diameter	36mm
Frame material thickness	0.4mm
Frame body dimension	62mmx62mmx183mm (LXWXH)
Frame cap	92mmx5mmx92mm (LxWxH)
Frame mesh	62mmx19mmx62mm
Wheel discharge diameter	14mm
Wheel design speed	2957rpm
Wheel design operating temperature	21.1 degree Celsius
Electric motor design speed	2970rpm at 50Hz
Electric motor design Horsepower	50Hp
Electric motor design operating temperature	40 degree Celsius Max

Before the modification, the following were obtained from blower manufacturer as the design specifications:

- RPM: 2957 (approximated to 3000)
- Static Pressure: 70 inches of water or 17Kpa
- Airflow rate: 1860 CFM or 0.96115 m^3/s

Given that impeller diameter, D = 1.2m and Impeller speed, N=2957RPM

The tip speed, $u = \frac{\pi DN}{60} = \frac{\pi \times 1.2 \times 2957}{60} = 185.818 \frac{m}{s} \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots$ (1)

By simplified Euler's turbomachinery equation,

$$\triangle P = \eta \rho \psi u^2$$

Assuming an efficiency of 70% and air density of 1.2kg/m^3 and a head coefficient of 0.6,

$$\triangle P \ = \ 0.7(1.2)(0.6)(185.818)^2 = 17.4\ KPa$$

Assuming an efficiency of 70% without a head coefficient and air density of 1.2kg/m^3

$$\triangle P \ = \ 0.7(1.2)(185.818)^2 = 29KPa \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots . (2)$$

$$Airflow\ rate\ Q = L \times B \times V = 0.11 \times 0.14 \times 185.818 = \frac{2.862m^3}{s} \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots \dots (3)$$

From simulation, Q= $\frac{2.862m^3}{s}$ and $\triangle P = 29KPa$ are achieved when N=4700RPM and the speed can still be reduced.

Haven gotten the required specifications from the manufacturer, a steady state CFD analysis using the SolidWorks simulation module was performed to predict the distribution of airflow throughout the system, as well as to analyse static pressure rise and velocity contours, which are critical for understanding the performance characteristics of the blower. A coupled CFD-thermal simulation was executed with the objective of studying the effectiveness of cooling on the turbine exhaust frame, thereby providing insights into thermal management strategies. While varying blade angles, impeller speeds, and flow rates were analysed to evaluate the optimal configuration for maximum efficiency and performance of the blower system. In this project, a performance evaluation was conducted under specific climatic conditions representative of the WRPC environment, considering variables such as temperature and humidity to assess their impact on the blower's operational efficiency. Figure 7 below shows the simulation of the original state when running to see the situation of the turbine.

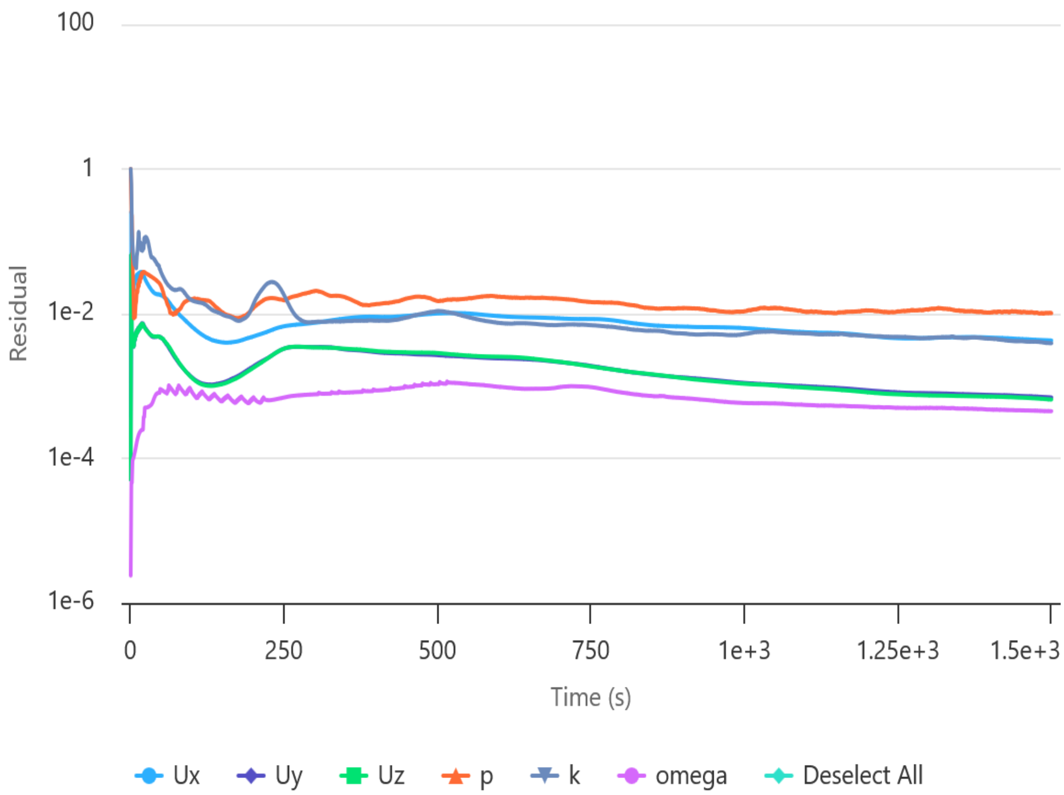

Figure 7: Goal residual plot of the simulation of the original state

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

The simulation presented an output pressure of 22.4KPa (Gauge pressure)

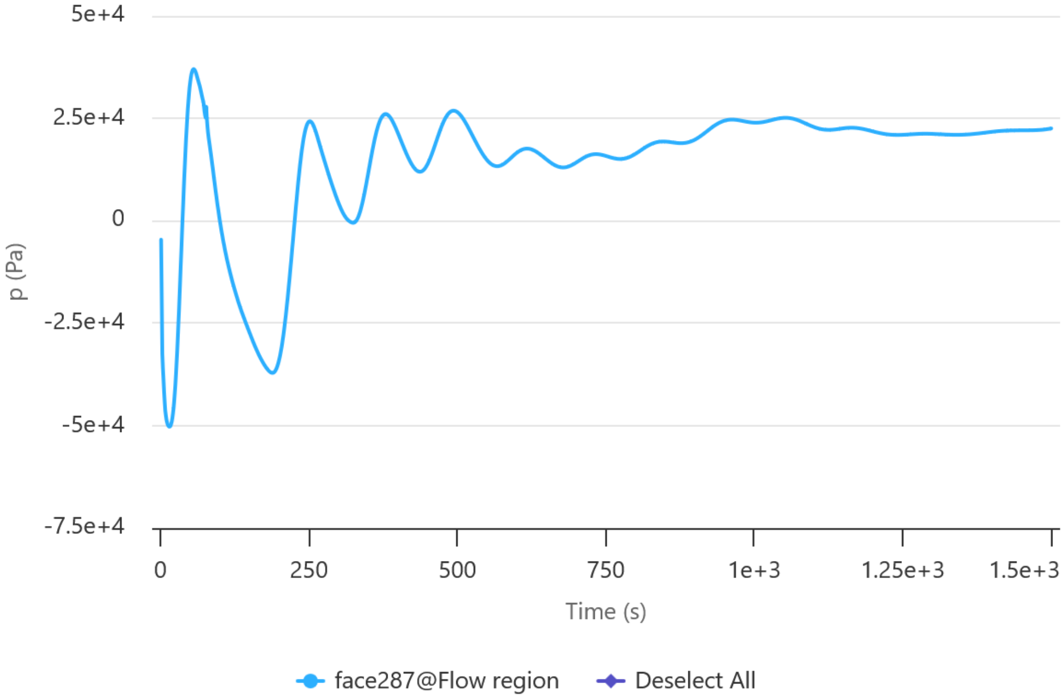

Figure 8: Pressure convergence of the simulation

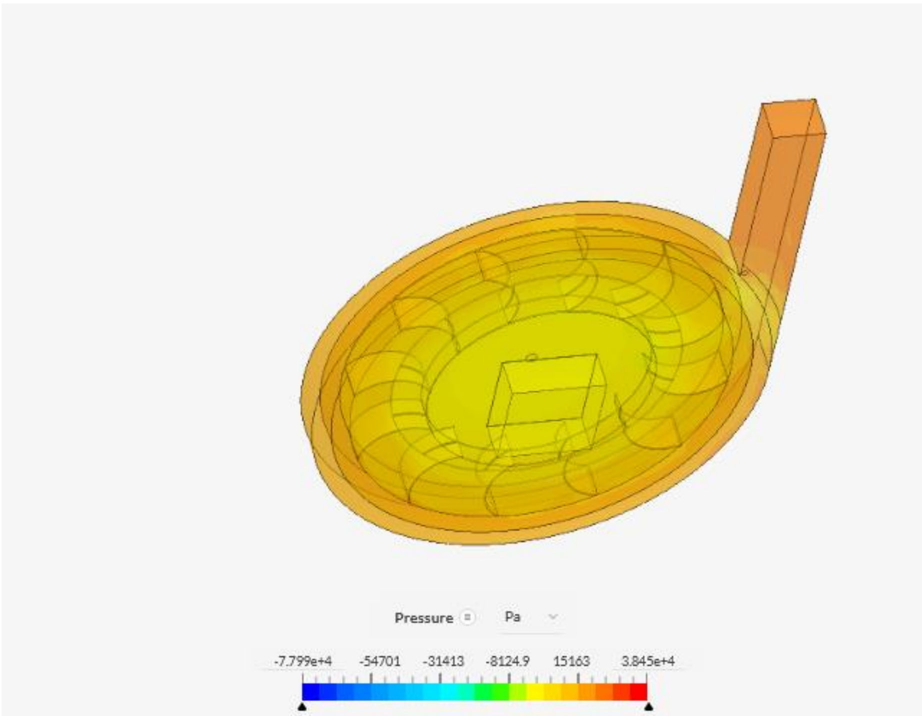

Figure 9: Existing blower model

The theoretical calculation aligns excellently with the manufacturer's specification of 17 kPa. This validates our modeling approach. The simulation result of 22.4 kPa, however, suggests higher than expected efficiency at the operating point. The simulation has idealized a few things and that explains why the output pressure is higher than the manufacturer’s value. For simulation purposes, we stick to the 22.4KPa so that even after losses, we will still be able to achieve the manufacturer’s figure.

The Blower Casing, fluid region model and Impeller was further assembled in SolidWorks assembly environment and imported into Ansys 2023 for analysis. Using Ansys Fluent, the mesh set up module was used to discretize the designed model using

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

a mesh element size of 0.87mm, inflation was further applied on the wall to account for boundary layer conditions. The model was further assigned named selections of Fluid Inlet, Fluid Outlet, Rotating region. Initial boundary conditions include:

- Inlet Pressure: 1 atm
- Volume flow Outlet: 0.96115 m^3/s
- Initial angular rotation of 3000RPM
- Initial Temperature of 308.88K

The following secondary boundary conditions were used

- Wall Condition-Adiabatic
- Roughness-0 Micrometre
- Flow Type- Turbulent
- Turbulence Intensity -2%
- Turbulence length- 0.00232m
- Pressure Based solver
- An MRF zone with a clearance ratio of 1.1 about the impeller.

The calculation was set to allow iterations until all goals were achieved and converged regardless of the number of iterations required

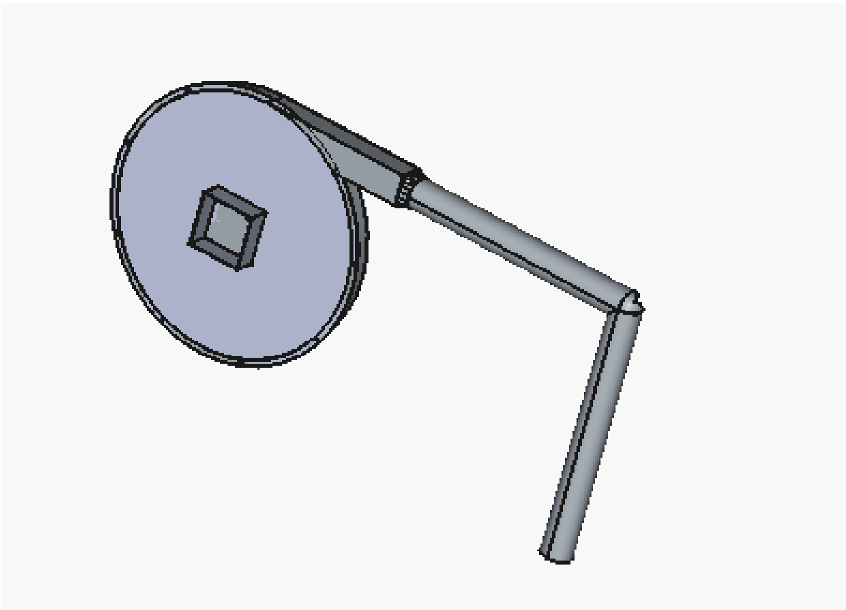

Figure 10: The assembly of the status quo of the blower

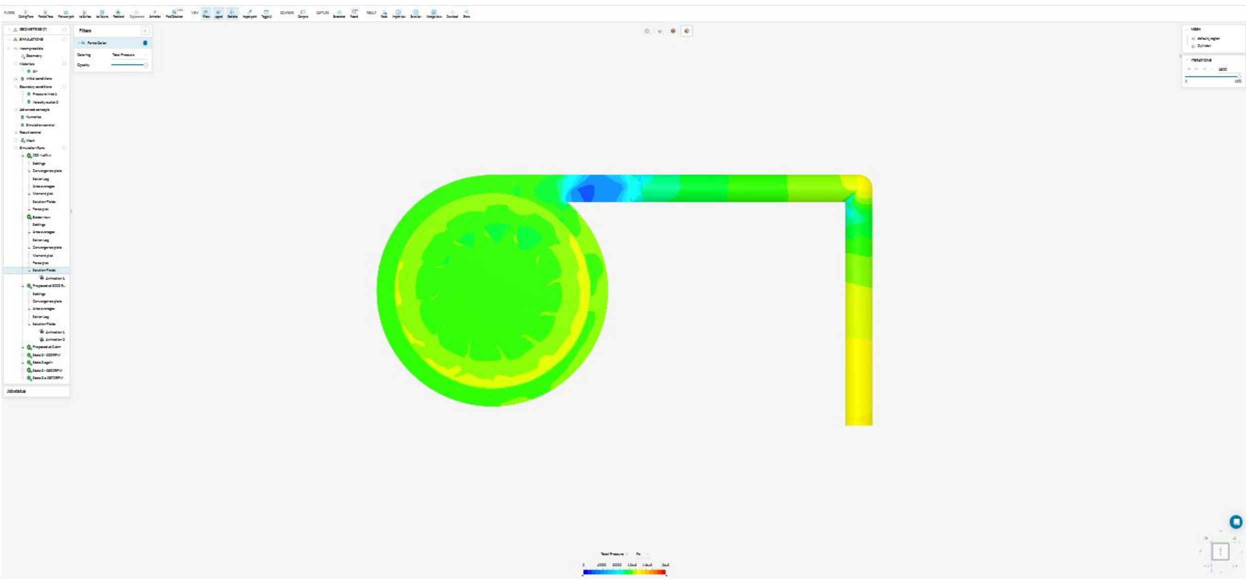

Figure 11: Pressure distribution across the blower

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

3.1 First Modification

To account for the piping system for the fluid horizontal and vertical travel, a right-angle pipe with a circular cross-section and internal diameter 140mm is coupled to the outlet.

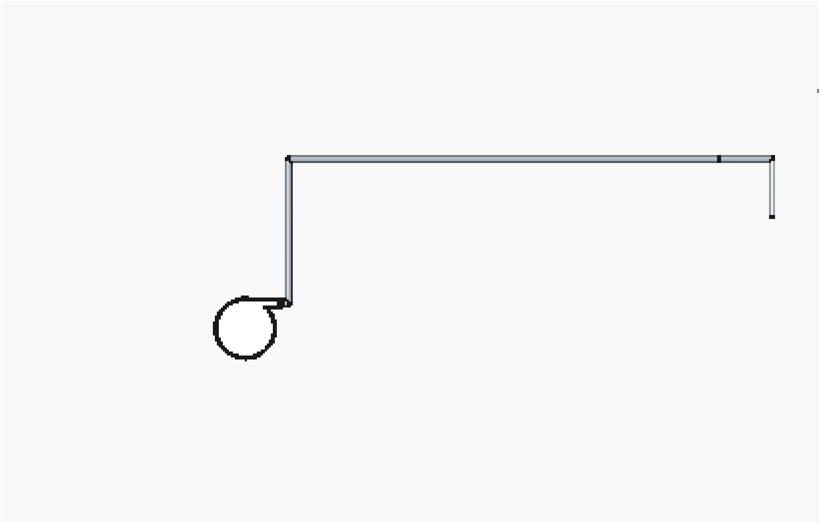

Figure 12: The first modification of the blower

Using Mesh quality check, the quality of the mesh was ascertained and validated using metrics like aspect ratio, element quality and orthogonal quality. The simulation set up module was then used to assign appropriate boundary conditions. The initial temperature was selected from the average of the last nine days which the temperature samples were taken from since the data for the first day represented an outlier. The average temperature for Morning, Afternoon and Evening of the nine samples is 308.88K, 321.38K and 318.08K. The lower temperature gas requires more rotational work; hence the lower temperature was used as inlet temperature to reflect the effect of temperature on the numerical analysis. Since the goal of the simulation is to determine the angular rotational velocity to deliver a static pressure of the 17 Kpa at the outlet, a simulation goal of pressure outlet is set while the rotational velocity varies in steps of 500 until the desired pressure outlet is achieved.

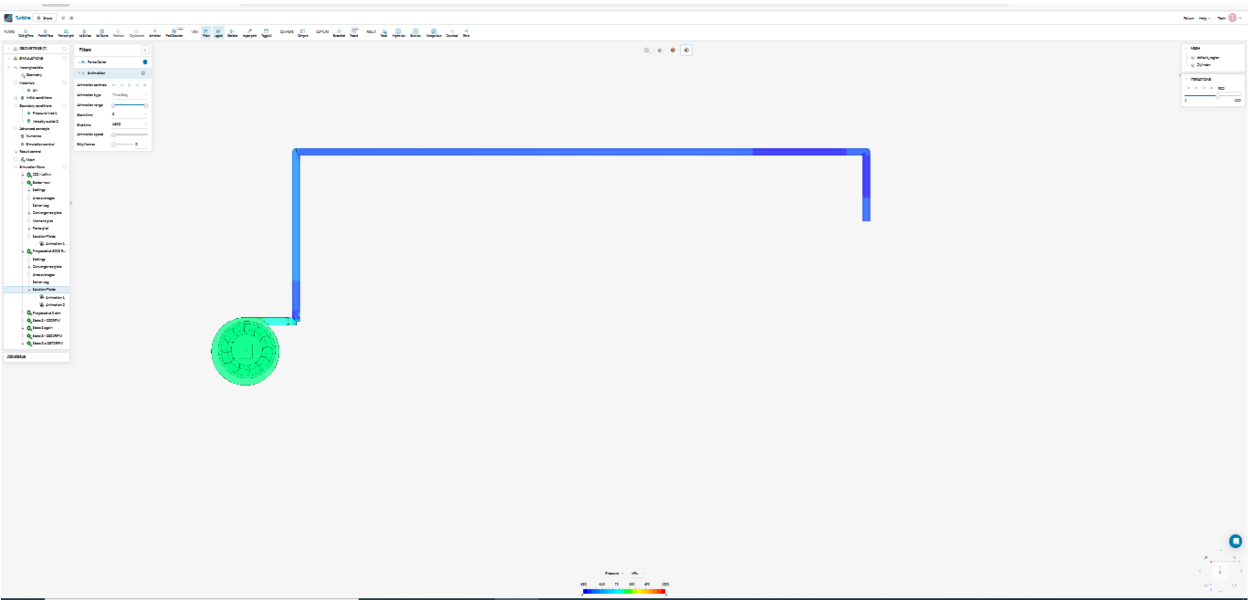

Figure 13: Pressure plot across the first modification

The first modification was simulated at 3000RPM (other conditions kept constant). The pressure contour plot shows an average outlet pressure of just about 0kPa. There was a sharp pressure drop observed at the coupling between the blower and the piping system.

3.2 Second Modification

Another modification was done to the piping system to remove the angular coupling and the vertical travel.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

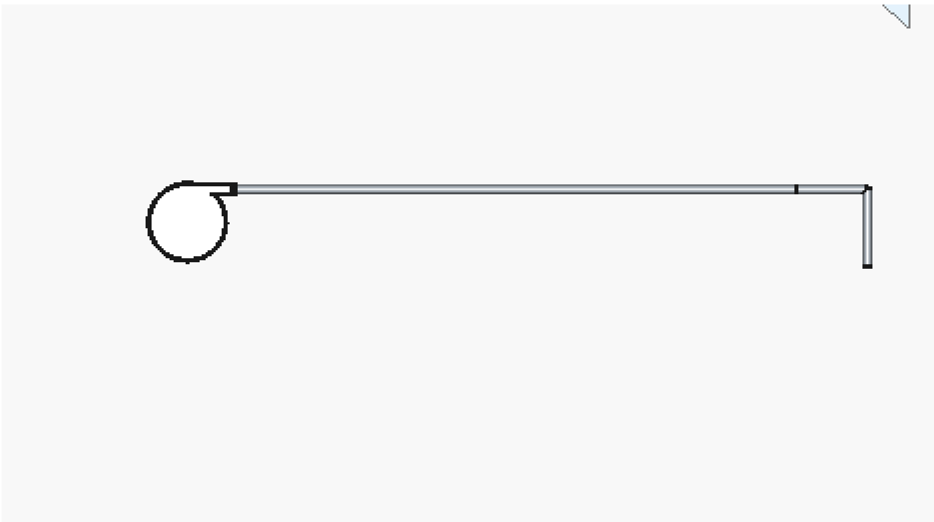

Figure 14: The second modification of the blower

$$\triangle P = KN^2D^2 \ldots\ldots (4)$$

$$\frac{\boldsymbol{D2}}{\boldsymbol{D1}} = \frac{\boldsymbol{N1}}{\boldsymbol{N2}} \ldots\ldots (\mathbf{5})$$

If we move from 4700RPM to 3500RPM

$$\frac{D2}{1.2m} = \frac{4700rpm}{3500rpm} \quad D2 = \frac{4700 \times 1.2}{3500} = 1.611m \ldots\ldots (6)$$

Approximately, if we increase the diameter of the impeller to 1.611m, we will match the same pressure, but the flow rate will increase.

$$\frac{Q2}{Q1} = \frac{N2}{N1} \times (\frac{D2}{D1})^3 = \frac{3500}{4700} \times (\frac{1.611}{1.2})^3 = 1.80 \ldots\ldots (7)$$

$$Q2 = 1.80Q1 = 1.80 \times 2.862 = \frac{5.157m^3}{s} \ldots\ldots (8)$$

From the above, the flow rate will increase by 80%

3.3 Model workflow

The design and simulation of a gas turbine exhaust frame blower using SolidWorks is presented according to the process flow below: Figure 7 shows the flow chart of the system.
The details of the system are highlighted below:

- Step 1: Define input parameters - Precisely record fluid (air/water), temperature and pressure (to derive ρ and μ).
- Step 2: Gather reference data & test conditions
- Step 3: Geometry preparation (CAD) - Clean imported geometry: remove tiny edges, close gaps, simplify fillets that force excessively fine mesh.
- Step 4: Mesh setup - Choose mesh type appropriate to solver and geometry. Also, agree on the boundary layer.
- Step 5: Boundary conditions & physical models, solver & numerical settings
- Step 6: Run simulation, then validate.
- Step 7: Refinement loop (if mismatch)
- Step 8: Post-processing & rep.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

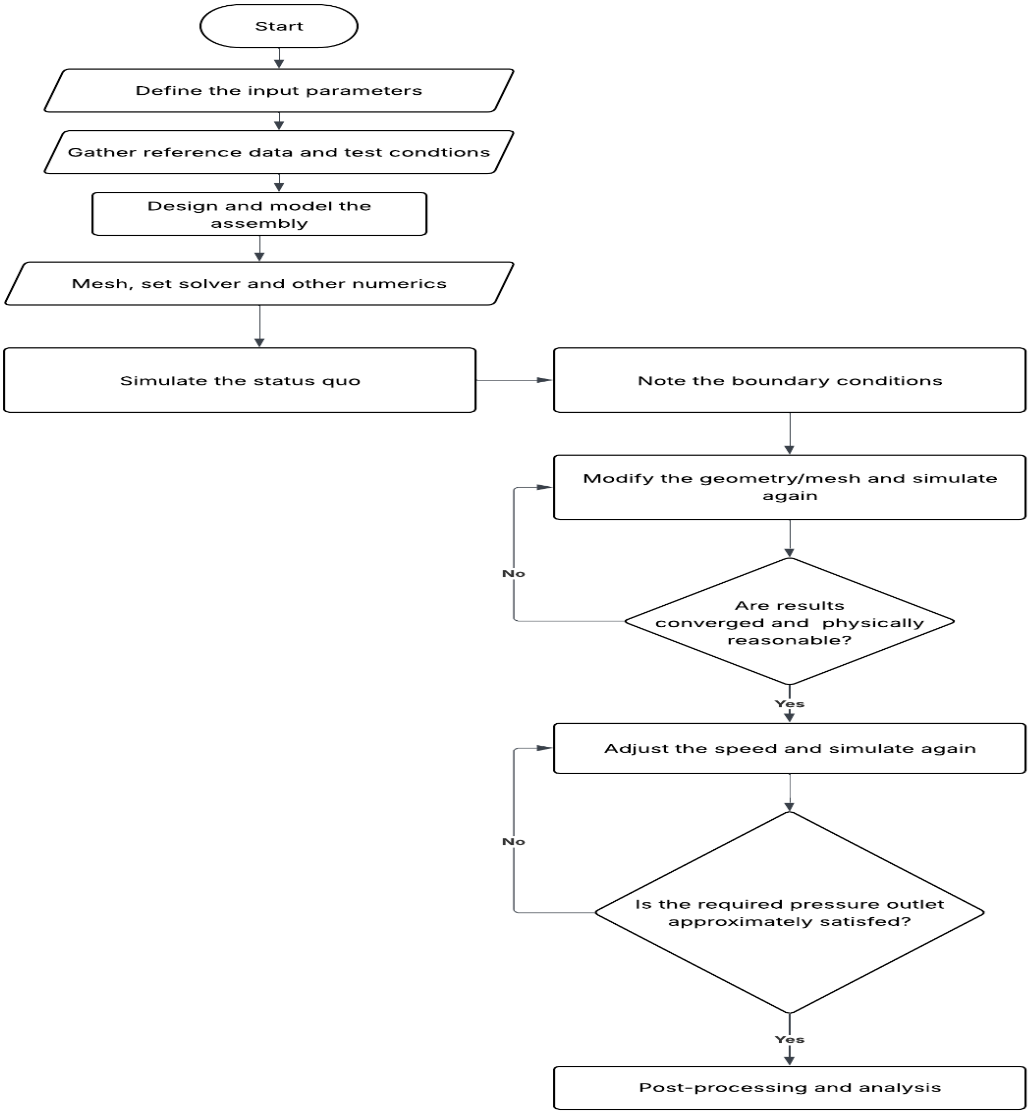

Figure 15: Model process workflow

4. RESULTS AND DISCUSSIONS

The angular velocities considered were 3000rpm, 3500rpm, 4000rpm, 3600rpm the output results for each angular velocity containing other simulation goals and Pressure contour plot (Minimum and Maximum) considered are highlighted below. Since the goal of the analysis is to understand the optimal angular velocity, there was no need to consider other parameters like velocity, temperature and general fluid behavior within the system.

4.1 Case 1

Angular velocity of 3000rpm, other parameters kept constant

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

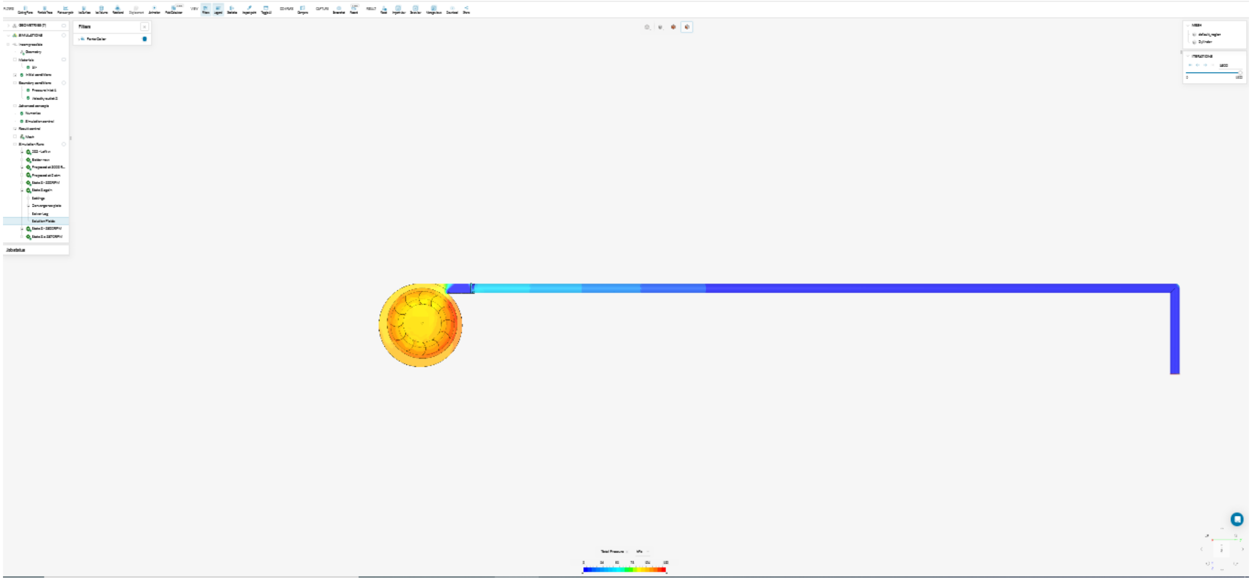

Figure 16: Pressure contour plot for angular velocity of 3000rpm

4.2 Case 2

Angular velocity of 3500rpm, other parameters kept constant.

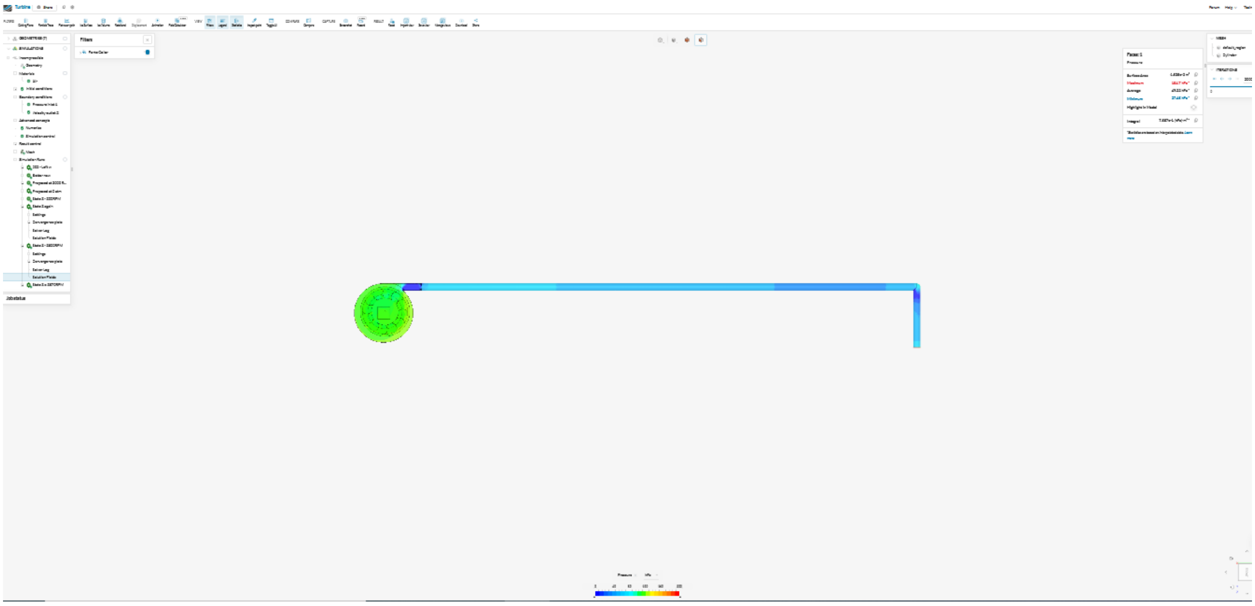

Figure 17: Pressure contour plot for angular velocity of 3500rpm

4.3 Case 3

Angular velocity of 4000rpm, other parameters kept constant.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

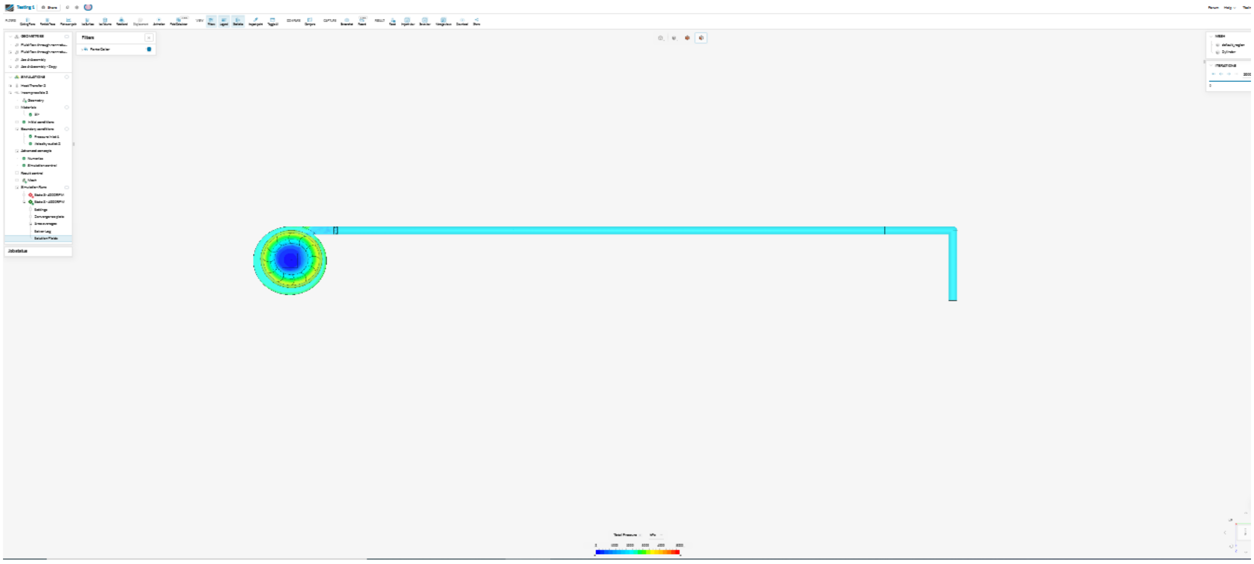

Figure 18: Pressure contour plot for angular velocity of 4000rpm

4.4 Case 4

Angular velocity of 3600 rpm, other parameters kept constant.

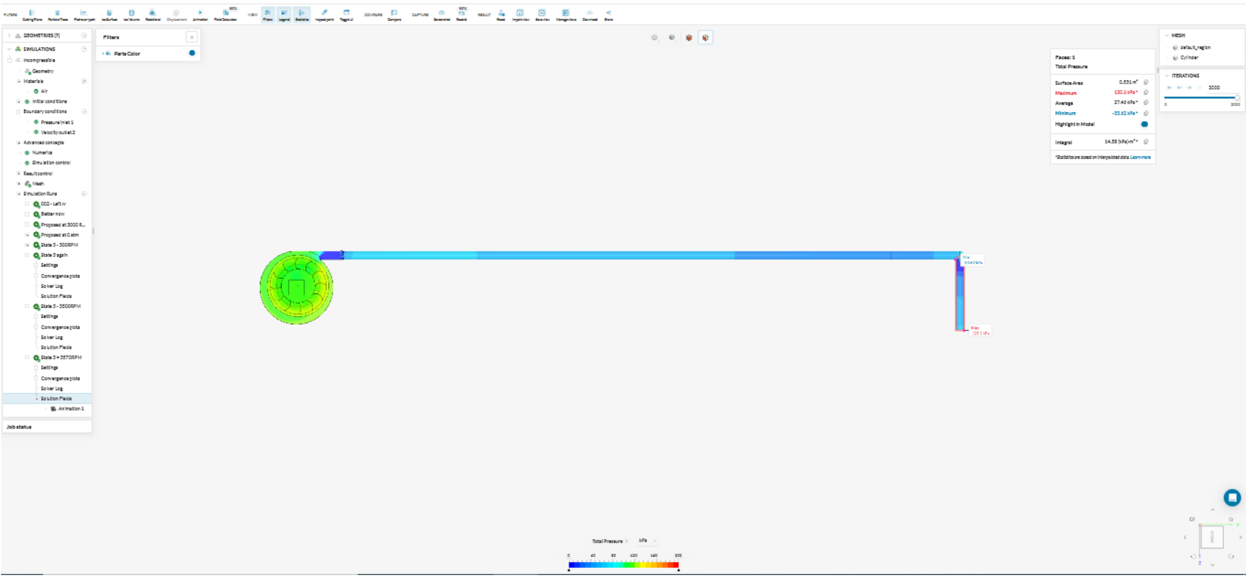

Figure 19: Pressure contour plot for angular velocity of 3600rpm

The analysis of pressure distribution at 3000RPM, 3500RPM, 3600RPM and 4000RPM reveals the fundamental energy conversion process within the blower. As depicted in the pressure contour plot, a distinct low-pressure zone is established at the impeller eye, which drives the suction of ambient air into the system. As the air is entrained by the rotating blades, its velocity and kinetic energy increase significantly. Upon discharge from the impeller tip into the volute casing, the fluid decelerates, causing a conversion of kinetic energy into potential energy, manifested as a rise in static pressure. The simulation demonstrates a smooth and continuous pressure gradient along the volute's spiral path, culminating in the highest pressure at the discharge outlet. This steady pressure rise, with minimal evidence of localized pressure drops or stagnation zones, indicates an efficient energy conversion process and validates the geometric design of the impeller and volute. The results obtained from case 1 to case 4 are summarized in the table below, with the pressure trend across the angular velocities considered shown in Figure 20

Table 2: Simulation Results

Angular Velocity(RPM)	Minimum Pressure(kPa.)	Maximum Pressure (kPa.)	Average Pressure(kPa.)
3000	71.21	139.5	101.9
3500	73.55	154.7	113.92
3600	77.54	174.03	120.1
4000	93.03	271.8	235.8

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

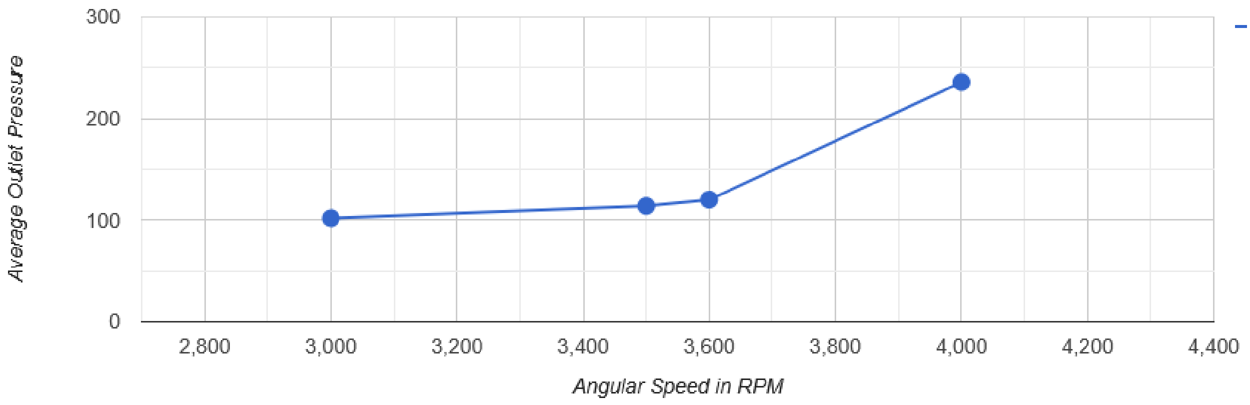

Figure 20: Outlet Pressure vs Angular Velocity

The velocity vector plots provide a complementary perspective, detailing the fluid's path and speed throughout the blower. The simulation shows that air enters the impeller eye axially and is smoothly turned into the radial direction by the curved blades. The velocity of the air increases substantially as it travels from the blade heel to the tip, reaching its maximum value just as it exits the impeller. This high-velocity discharge is a direct result of the energy imparted by the rotating impeller. Within volute, the velocity vectors show a gradual decrease in magnitude as the cross-sectional area of the scroll increases, consistent with the principle of diffusion. Importantly, the flow trajectories are largely uniform and well-aligned with the volute's curvature, indicating that the design successfully minimizes chaotic turbulence and energy-dissipating eddies. The absence of significant flow separation from the blade surfaces further confirms the appropriateness of the 35° inlet and 26° outlet blade angles, which are critical for maintaining attached flow and achieving high aerodynamic efficiency.

5. Conclusions

This paper has presented the designs and simulation of a high-efficiency exhaust frame air blower to address the critical operational challenge of gas turbine overheating at the Warri Refining and Petrochemical Company. By integrating fundamental principles of turbomachinery with advanced CFD simulation tools, a solution was developed to meet the specific demands of a tropical industrial environment. The design process focused on creating a system that is not only functional and efficient but also prioritizes safety. Key findings include.

1. In comparison to the original set up, the first modification achieves the same output parameters with a higher angular velocity requirement of the about 4000 RPM.
2. In comparison to the original set up, the second modification achieves the same output parameters with a higher angular velocity requirement of the about 3600 RPM.
3. This speed in the second modification is not expected to give many problems since it is close to the current working speed. However, extra care must be taken about the mounting of the blower. A strong scaffold with a wide base should be considered for mounting the blower.
4. An angular velocity of 4000 RPM in the first modification for a 1.2m diameter impeller is achievable but comes with risks of concentrated stress, and torque. This can be mitigated if the blower is designed with specialized materials for handling the stress.
5. It is expected that the diameter can be scaled in the first modification using similarity affinity laws to deliver the same target pressure at a lower speed of about 3000 rpm, delivering the same pressure but at a higher flow rate.
6. It is to be noted that the torque at 5000RPM dropped below that of 4500RPM. A simple explanation is that failure is expected to occur at or around that speed. A speed of 4000 RPM is more achievable and looks safer to pull off for the first modification

DISCLAMER (ARTIFICIAL INTELLIGENCE)

Author(s) hereby declares that NO generative AI technologies such as Large Language Models (CgatGPT, COPILOT, etc) and text to image generators have been used during writing or editing of this manuscript.

ACKNOWLEDGEMENTS

Firstly, my gratitude goes to God Almighty for the grace and provision all throughout the duration of this program. Special thanks to my amiable Project Supervisors, Engr. Prof. Ashiedu Festus Ifeanyi and Engr. Prof. Emagbetere Eyere for their support, guidance and supervision of this project work. I acknowledge all teaching and non-teaching staff of the Department of Mechanical Engineering, Federal University of Petroleum Resources, Effurun.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 15 Issue 01 , January - 2026

Also, I acknowledge my beloved wife, Mrs. Hassan, my Children, for the support, encouragement and for availing me the peace of mind to concentrate on this work to the end. Lastly, I want to acknowledge my parents, for their support and prayer.

AUTHORS' CONTRIBUTION

Author '1' designed the study, developed the model for the optimized relocation of the exhaust frame blower, simulated the design, and wrote the first draft of the manuscript. 'Author 2' managed the analysis of the study, managed the literature searches. 'Author 3' wrote the second draft of the manuscript and all the authors read and approved the final manuscript."

REFERENCES

[1] Al-Dabooni, S. J. (2022). Using Fuzzy Inference System in Gas Turbine to Overcome a High Exhaust Temperature Problem. *Journal of Petroleum research and study*, Vol 12, no 1.
[2] Azzawi, I. D. (2023). Design and characterizing of blower wind tunnel using experimental and numerical simulation. *Proceedings Of The Institution Of Mechanical Engineers, Part G: Journal Of Aerospace Engineering,*, 237(15), 3582–3596.
[3] C. Keerthana, D. P. (2019). Design and Analysis of a Combustion Chamber in a Gas Turbine. *International Journal of Engineering Research & Technology*, ISSN: 2278-0181.
[4] Cai, Q. (2019). Blower capable of conducting self-physical cooling.
[5] G. Saibabu, V. R. (2021). MODELING AND STRUCTURAL ANALYSIS OF COMPOSITE CENTRIFUGAL BLOWER USING FEM. *Open access International Journal of Science and Engineering* , Volume 6 || Issue 2 || pp 84.
[6] Han, J.-C. (2018). Advanced Cooling in Gas Turbines Max Jakob Memorial Award Paper. *Journal of heat transfer*.
[7] Hara, F. &. (2020). Blower and air conditioner.
[8] Hassan L. Bagudo, A. I. (2025). Thermal Management and Reliability Enhancement of GE Gas Turbine Exhaust Frame Blowers through Relocation and Design Simulation. *Journal of Engineering Research and Reports*.
[9] Herman, A. (2018). Well optimization using downhole blower system.
[10] Hoffman, L. C. (2007). Blower assembly with integral injection molded suspension mount. .
[11] J. Han. (2018). Advanced Cooling in Gas Turbines 2016 Max Jakob Memorial Award Paper. *Journal of Heat transfer*.
[12] J.D Anderson. (2012). Computational fluid dynamics . *Tata Mc-Grew Hill Edition*.
[13] Jang, C.-M. &.-J. (2014). Performance Enhancement of 20kW Regenerative Blower Using Design Parameters. *International Journal of Fluid Machinery and Systems*, 7(3), 86–93.
[14] Kreidler, J. J. (2017). Blower, electric machine and associated method.
[15] L. Silva. (2017). Analyzing Burn Out and Random Trips at Starting: Induction Motors Driving Main Blowers. *IEEE Industry Applications Magazine.*
[16] Lakshya Kumar, D. B. (2019). Design and CFD Analysis of a Multi Stage Axial Flow Compressor. *21st Annual CFD Symposium, CFD Division.* Bangalore: Aeronautical Society of India.
[17] Lichtblau, L. (1988). Blower for delivering cooling air to internal combustion engines.
[18] Nasrollahi, S. &. (2024). Effects of Various Air Flow Modification Techniques on Design and Development of a Multi-fan Blower Type Open-Section Civil Wind Tunnel.
[19] P. Laohasongkram, S. L. (2007). Solving Problem of Gas Turbine and HRSG Trip from Hydraulic Damper Closing in Combined Cycle power plant . *International Conference on Control, Automation and Systems*.
[20] P. Srinivasulu, N. S. (2022). Design and Simulation of Centrifugal Blower Using with Different Composite Materials . *IJARIIE*, Vol-8 Issue-5, 2395-4396.
[21] Peng, W. (2022). Design and Characterizing of Blower Wind Tunnel Using CFD.
[22] Rini, D. P. (2015). Centrifugal Blower for Personal Air Ventilation System (PAVS) - Phase 1.
[23] Roger L. Davis, J. Y. (2006). Prediction of Compressor Stage Performance from Choke Through Stall. *Journal of Propulsion and Power*, Vol 22, no 3.
[24] Song, S. B. (2005). Blower and design method of discharge port thereof.
[25] T.S Chowdhury, F. T. (2023). A Critical Review on Gas Turbine Cooling Performance and Failure Analysis of Turbine Blades . *International Journal of Thermofluids*.
[26] Thomas, C. S. (2019). Optimization Fuzzy Controller Parameters for Soot Blower in Power Plant. . 3(1), 180–185.
[27] Vincenzo Castorani, D. L. (2019). Design Optimization of Customizable Centrifugal Industrial Blowers for Gas Turbine Power Plants. *Computer-Aided Design & Applications*, 16(6),1098-1111.
[28] Zhang, X. Y.-x. (2024). Numerical Investigation on the Effects of Gap Circulating Flow on Blower Performance under Design and Off-Design Conditions. . *Energies*, 17(15),.