Teknomekanik, Vol. 9, No. 2, 2026, pp. 203-217
e-ISSN: 2621-8720 p-ISSN: 2621-9980

CFD-based Taguchi optimization of impeller geometry to improve centrifugal fan efficiency

Delima Yanti Sari[1*], Bagas Santoso[1], Hendri Nurdin[1], Hastuti[2], Rifelino[1], Fitrah Qalbina[1], Tsung-Liang Wu[3], Dani Harmanto[4]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Padang, **Indonesia**
[2] Department of Electrical Engineering, Faculty of Engineering, Universitas Negeri Padang, **Indonesia**
[3] Department of Mechatronics Engineering, National Kaohsiung University of Science and Technology, **Taiwan**
[4] Department of Aeronautical Engineering, De Montfort University, **United Kingdom**

*Corresponding Author: delimayanti@ft.unp.ac.id
Received: 26 January 2026; *Revised:* 01 May 2026; *Accepted:* 15 May 2026

Cite this https://doi.org/10.24036/teknomekanik.v9i2.53772

Abstract: Centrifugal fans play a significant role in industrial ventilation systems. Their performance is affected by aerodynamic losses such as flow separation and non-uniform pressure distribution, thereby reducing the overall efficiency. Since design parameters influence the fan efficiency, design parameter optimization becomes one option for addressing this issue. Some optimization methods involve high computational cost and complex procedures. This raises the necessity for a more efficient and systematic optimization procedure. The aim of this study is to propose an integrated approach which involves Computational Fluid Dynamics (CFD) and the Taguchi method to improve the performance of a centrifugal fan. CFD is used to evaluate the performance of different design combinations, while the Taguchi method is used to optimize design parameters. The investigated design parameters are the inlet blade angle (β1), the outlet blade angle (β2), the number of blades (n), and the flow rate (Q). Each of the design factors has three levels, therefore, an L9 orthogonal array was utilized as the design of experiments. Analysis of variance (ANOVA) is used to determine their relative significance. The results show that the optimal combination of design parameters increase the efficiency from 39.79% (the reference) to 63.26%. The CFD simulations for the optimal combination exhibit the improved flow behaviour, which explains the enhanced efficiency. The results show the feasibility of the proposed method for improving the performance of the centrifugal fan.

Keywords: centrifugal fan; design configurations; HVAC systems; impeller geometry optimization

1. Introduction

Centrifugal fans play a significant role in industrial ventilation and HVAC systems due to their ability to deliver high airflow under varying pressure conditions [1]. The increase in global energy demand and regulations on efficiency make improving fan performance a priority. Several studies reveal that the impeller geometry significantly affects the efficiency of centrifugal fans [2], [3]. In industrial practice, many centrifugal fans still operate at low efficiency. This is caused by the aerodynamic losses which arise due to suboptimal impeller geometry. This low efficiency imposes a higher operational energy consumption. The design parameters, such as inlet blade angle (β1), outlet blade angle (β2), number of blades (n), and flow rate (Q), are identified as primary geometric parameters that significantly influence the fan performance [4], [5]. However, these parameters do not act independently. Their interactions collectively affect the flow behavior and efficiency. For example, inappropriate inlet blade angles might cause flow separation at the entry. On the other hand, excessive or insufficient blade numbers may increase blockage effects or reduce flow guidance effectiveness. The outlet blade angle is found as a parameter which influences pressure rise and flow direction. However, several studies found results that are inconsistent about its optimal value which depends on operating conditions and design configurations [6]. Generally, incorrect combinations of these parameters may cause recirculation zones, increase turbulence and distribution of non-uniform pressure. Eventually, they contribute to significant aerodynamic losses and decrease the efficiency [7], [8]. In addition, some previous studies do not clearly distinguish between geometric design variables and operating conditions, which may lead to biased interpretations in statistical analyses [9].

Several studies perform centrifugal fan optimization [10], [11], [12], [13]. Some advanced optimization methods, namely response surface methodology, genetic algorithm, and particle swarm optimization, have been used in the design of centrifugal fans. However, these methods generally require a large number of Computational Fluid Dynamics (CFD) and high complexity. This induces a high computational cost and longer processing time. Therefore, an efficient optimization method, which systematically captures multi-parameter interactions, is required. The aim of the present study is to improve the efficiency of the centrifugal fan by optimizing design parameters through an integrated method combining CFD and the Taguchi method. The Taguchi method is used for optimization because it reduces computational cost while maintaining the ability to effectively evaluate multiple design parameters. This method has been introduced as a structured design-of-experiments approach that uses orthogonal arrays to identify significant parameters with a minimal number of simulations [14], [15], [16]. To evaluate the performance of different design configurations, CFD was used as a numerical tool, while the Taguchi method was used to find the optimal parameters. CFD has been commonly used to analyze the flow characteristics of centrifugal fans. By using CFD, the evaluation of velocity fields, pressure distribution, and aerodynamic losses are obtained [17], [18].

This study proposes a novelty in the formulation of an optimization framework, where geometric design variables, namely angle of inlet blade (β_1), angle of outlet blade (β_2), number of blades (n), and the flow rate (Q) are involved as design factors. The results of this study are expected to be useful for the development of high-efficiency centrifugal fans. This research is expected to contribute to the development of a more efficient method for multi-parameter optimization of centrifugal fans and to provide practical design knowledge from the integration of statistical and numerical methods.

2. Material and methods

2.1 Research method

This study involves CFD and Taguchi method as a structured optimization approach. The research methodology is illustrated in Figure 1. The Taguchi method is utilized to construct an L9 orthogonal array. This allows systematically evaluating multiple design parameters with a minimal number of simulation cases. For each parameter combination, CFD simulations are performed by using Ansys Student R1 2024 to predict efficiency, velocity, and pressure distribution. The CFD simulation results are further analysed by Taguchi to evaluate the significance of each design parameter and to obtain the optimal combination.

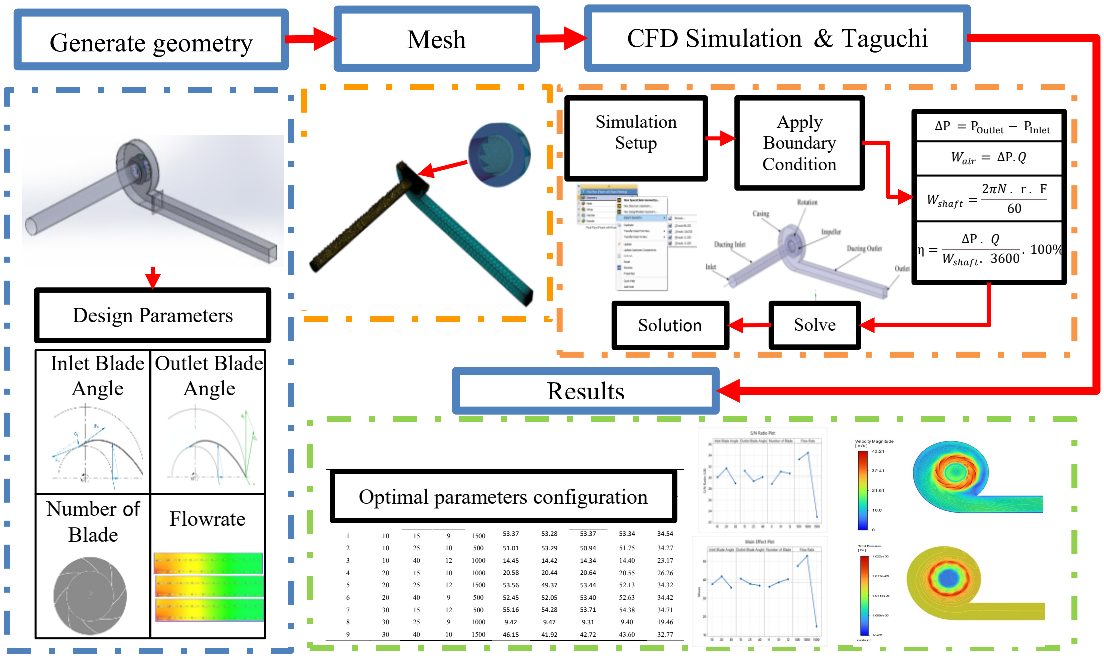


Figure 1. Flowchart

2.2 Fundamental formula

The performance of a centrifugal fan is characterized by several parameters, including efficiency, total pressure rise, and shaft power. The total pressure rise (ΔP) is the pressure difference between the outlet and inlet. It can be written as in Equation (1).

$$\Delta P = P_{total\ outlet} - P_{total\ inlet} \tag{1}$$

Another critical parameter used to quantify the fan's performance is air power. This is a measure of the actual energy transferred from the impeller to the moving air. It is usually computed as the product of the total pressure rise (ΔP) and volumetric flow rate (Q), as shown in Equation (2).

$$W_{air} = \Delta P.Q \tag{2}$$

In this expression, the pressure rise, measured in pascals (Pa), is a function of the volumetric flow rate Q, measured in cubic meters per hour (m3/h). The symbol W_{shaft} represents shaft power, measured in watts (W). Shaft power in turn is the mechanical energy transferred to the fan impeller. It is calculated by multiplying the angular velocity (ω, in radians per second) by the shaft torque (Nm), as presented in Equation (3).

$$W_{shaft} = \omega . T_{shaft} \tag{3}$$

Angular velocity (ω) can be expressed in terms of fan rotation speed (N) in revolutions per minute (rpm), as in Equation (4).

$$\omega = \frac{2\pi N}{60} \tag{4}$$

Physically, the torque of the shaft can also be calculated using the product of the radius at which the force is applied (r), and the tangential force applied on the shaft (F), as written in Equation (5).

$$T_{shaft} = r\,.\,F \tag{5}$$

By substituting the value of ω into the shaft force, it can be formulated as in Equation (6).

$$W_{shaft} = \frac{2\pi N\,.\,\mathrm{r}\,.\,\mathrm{F}}{60} \tag{6}$$

The total pressure rise includes both static and dynamic pressure. Furthermore, the efficiency of the fan (η), which is a key factor in the evaluation of the performance of the fan, is defined as the ratio of the useful power output, i.e., the fluid pressure energy, to the input power provided by the shaft of the fan, as shown in Equation (7).

$$\eta = \frac{W_{air}}{W_{shaft}}.100\% \tag{7}$$

From the obtained results, it can be derived as in Equation (8).

$$\eta = \frac{\Delta \mathrm{P}\,.\,Q}{W_{shaft}.\,3600}.100\% \tag{8}$$

2.3 Model construction

The geometry of this study is obtained from a commercially available industrial fan. The numerical model was created in SOLIDWORKS. The geometry of the centrifugal fan is presented in Table 1 and Figure 2.

Table 1. Geometry of a centrifugal fan

Parameter	Size
Diameter of Impeller (a), mm	287
Angle of Blade (b), degree	30º
Number of Blades	10
Width of Impeller (c), mm	95
Diameter of Inlet (d), mm	170
Outlet Size (e), mm x mm	153 x 130
Length of Inlet Channel (f), mm	1700
Length of Outlet Channel (g), mm	1900
Thickness of Casing, mm	5
Width of Blade, mm	3

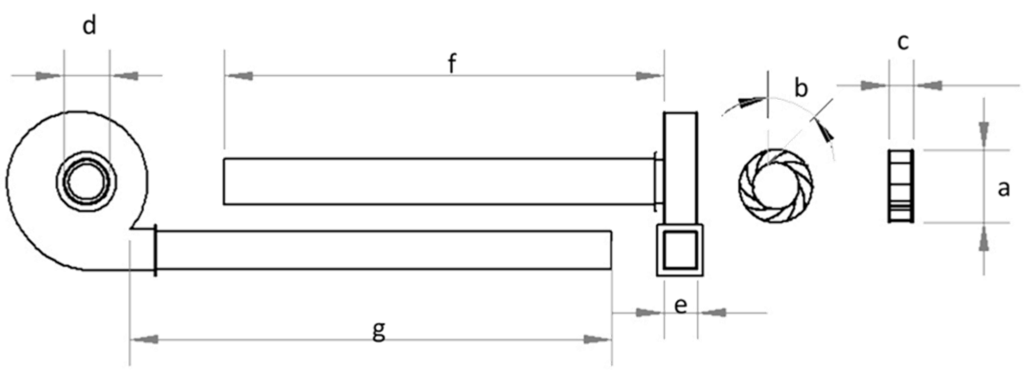


Figure 2. Geometry definition of the centrifugal fan

2.4 Boundary conditions

In this study, the CFD simulations are conducted under steady-state conditions. The flow is assumed to be incompressible, with negligible gravitational effects and constant fluid properties [19], [20]. A no-slip boundary condition is applied to all solid walls to accurately capture viscous effects near the surface. A velocity inlet is specified based on a fixed operating flow rate corresponding to the design condition, while a pressure outlet is set to atmospheric pressure, following standard practices in fan performance simulations. The impeller rotation is modelled using a Rotating Reference Frame (RRF) approach with a constant angular velocity of 301.069 rad/s, which is widely adopted for steady-state analysis of rotating machinery. For the turbulence model, the realizable k-ε model is used due to its robustness in predicting rotating and separated flows, as reported in previous CFD studies on centrifugal fans. Standard wall functions are used for near-wall treatment to ensure computational efficiency and numerical stability.

The simulations are performed using a pressure-based solver with second-order discretization schemes, and convergence is achieved when all residuals fall below 10^{-5}, which is consistent with established CFD practices. A schematic representation of the boundary conditions and computational domain are shown in Figure 3.

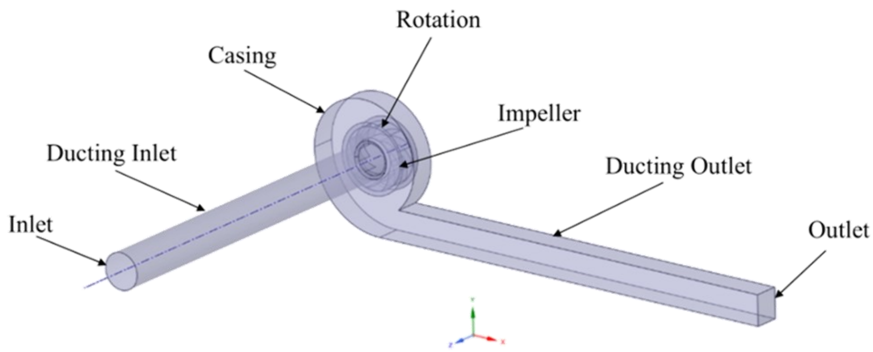


Figure 3. Boundary Conditions on the simulation model

2.5 Selecting the optimal mesh for CFD analysis

The validation of finite element modelling (FEM) is an important step in ensuring the reliability of simulation results obtained from such models [21], [22]. In the present study, validation is achieved by comparing the CFD simulation results with available experimental data reported in the literature [23], [24]. In addition, reference data obtained from previous studies are used as a comparative benchmark to ensure consistency of the numerical approach [25]. A mesh dependency (grid convergence) study was conducted using four different mesh densities (Mesh A–D) to evaluate the influence of mesh resolution on the simulation results. The results were compared with key performance parameters, particularly total pressure rise and efficiency. The sensitivity of the mesh was analyzed at several levels, as depicted in Table 2. According to the mesh sensitivity analysis adopted from [25], Mesh B shows the lowest error (5%), as shown in Figure 5, indicating the highest accuracy. It can be observed that the predicted values gradually approach a stable solution as the mesh density increases. The difference between Mesh C and Mesh D is relatively small, indicating that the solution is beginning to converge and is less sensitive to further mesh refinement. In addition, the simulation results were compared with reference data [25] to validate the numerical model. Among the tested meshes, Mesh B shows good agreement with the reference values while maintaining reasonable computational cost. Therefore, Mesh B was chosen as the optimal mesh for subsequent simulations, balancing accuracy and computational efficiency.

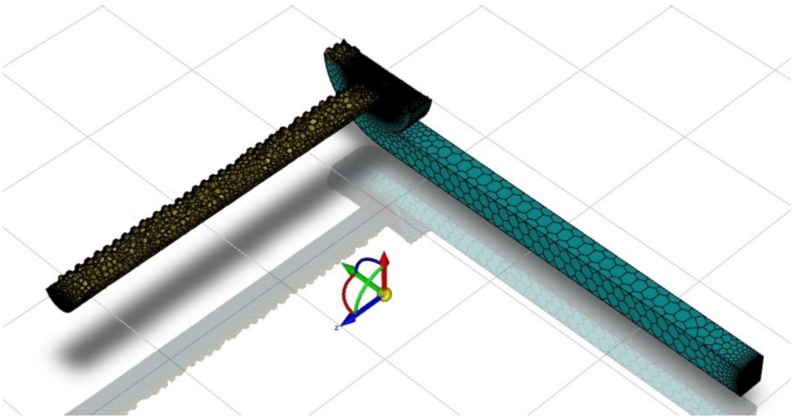


Figure 4. Computational mesh of the simulation model

Table 2. Mesh dependency test

Parameter	Reference [25]	Mesh A	Mesh B	Mesh C	Mesh D
Number of Mesh	591329	273439	563508	766890	993788
Pressure Inlet (Pa)	100384.88	100173.3	100161.7	100150.4	100133
Pressure outlet (Pa)	101338.42	101331.5	101331.6	101331.6	101331.6
Total Pressure Rise (Pa)	953.54	1158.2	1169.9	1181.2	1198.6
Air Power	52.97	64.25	64.95	65.58	66.55
Shaft Power	177.03	259.98	206.65	204.18	200.67
Efficiency (%)	29.92	24.71	31.43	32.12	33.16
Error Percentage to Reference [25]		17.41	5.05	7.35	10.83

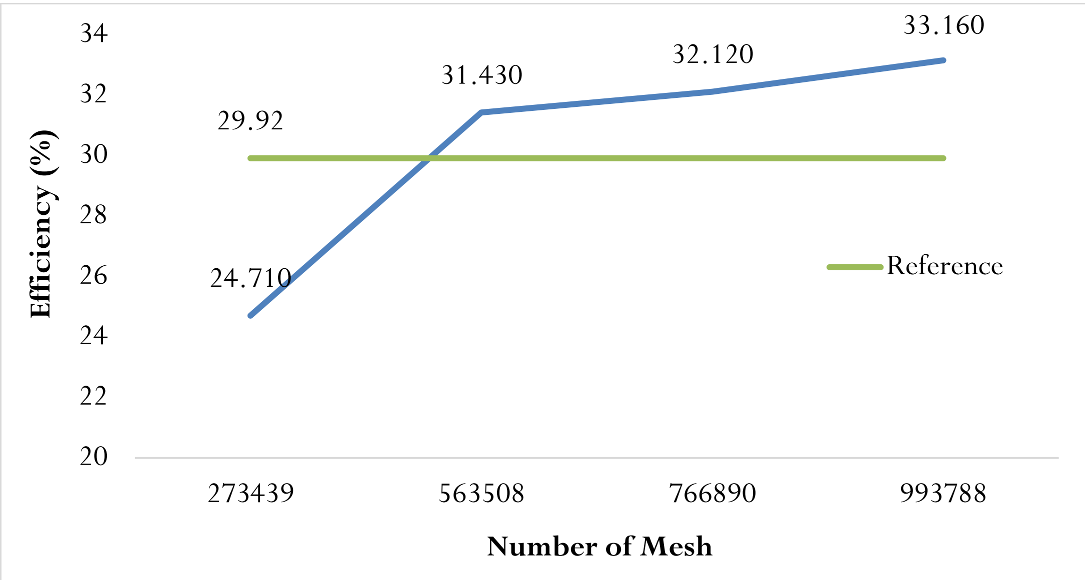
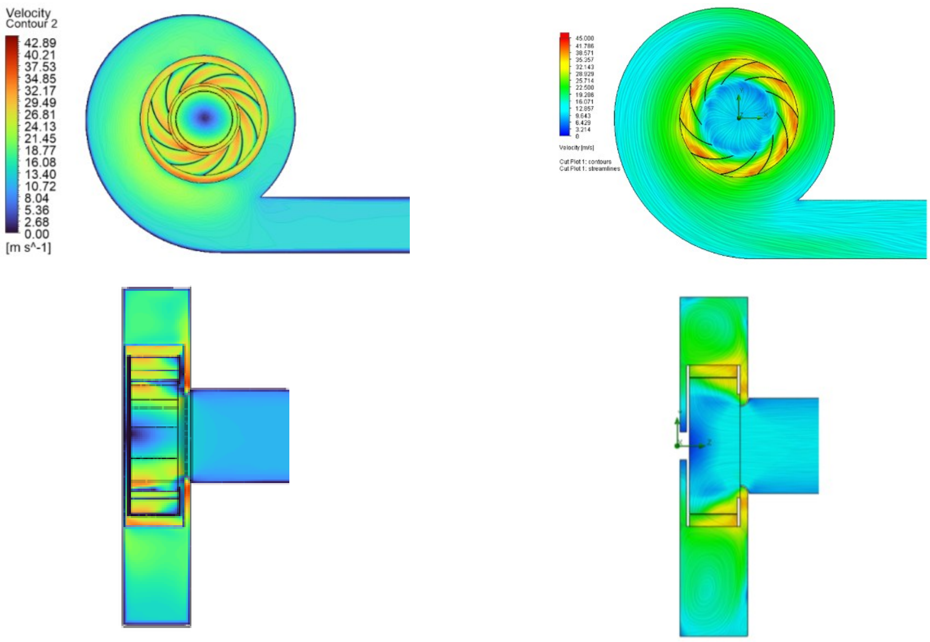


Figure 5. Mesh sensitivity analysis of fan efficiency



(a) (b)

Figure 6. Velocity Contour (a) Ansys and (b) Previous Study by Solidwork [25]

2.6 Determination of orthogonal array for impeller geometry variations

After validating the CFD results, optimization was conducted by using the Taguchi method. The Taguchi method was chosen for its ability to optimize several design parameters with a limited number of simulations without sacrificing the accuracy [26], [27]. Here, the L9 orthogonal array is used to evaluate three levels of four control factors, namely the inlet blades angle (β_1), outlet blades angle (β_2), number of blades (n), and flow rate (Q). The levels are presented in Table 3.

Table 3. Experimental design based on the Taguchi method

Number	Control Factor	Variation		
		Level 1	Level 2	Level 3
A	Inlet Blade Angle (β_1)	10	20	30
B	Outlet Blade Angle (β_2)	15	25	40
C	Number of Blades (n)	9	10	12
D	Flow Rate (Q) m^3/h	500	1000	1500

The Signal-to-Noise Ratio (S/N) is used to assess the stability and quality of the design parameters against undesirable variations, ensuring robust performance and optimal operation of the centrifugal fan under different conditions. For performance optimization, the "Larger-the-Better" approach is used in the S/N calculation, aiming to maximize the response (efficiency) [16]. The formula for calculating the Signal-to-Noise Ratio (S/N) for the *Larger-the-Better* characteristic is defined in Equation (9).

$$S/N = \pm 10\, log_{10}\left(\frac{1}{n}\sum_{i=1}^{n}\frac{1}{y_i^2}\right) \quad (9)$$

2.7 ANOVA

Analysis of variance (ANOVA) is used to evaluate the influence of each design parameter on the response variable (efficiency)[28]. The total variation of the response is decomposed into contributions from each factor using the sum-of-squares method. The total sum of squares (SST) is formulated as in Equation (10).

$$SST = \sum_{i=1}^{N}(y_i - \bar{y})^2 \tag{10}$$

y_i is the response value and $\bar{y}$ is the mean response. The sum of squares for each factor is calculated as in Equation (11).

$$SS_{factor} = \sum_{j=1}^{k} n_j(\bar{y}_j - \bar{y})^2 \tag{11}$$

$\bar{y}_j$ is the mean response at level j, and n_j is the number of observations at that level. The mean square (MS) is obtained by Equation (12).

$$MS = \frac{SS}{DoF} \tag{12}$$

while the F-value evaluates the strength of the factor compared to experimental error. The F-value is calculated as in Equation (13).

$$F = \frac{MS_{factor}}{MS_{error}} \tag{13}$$

Statistical significance is determined by the p-value. A value less than 0.05 indicates that the factor has a significant effect on the response. The p-value is calculated as in Equation (14).

$$\text{p} = \text{P}\ (F_{(df1,df2)} > F_{count}) \tag{14}$$

The contribution percentage indicates the relative influence of each factor on the response. The percentage contribution of each factor is expressed as in Equation (15).

$$Contribution(\%) = \frac{SS_{factor}}{SST}\ x\ 100\% \tag{15}$$

3. Results and discussion

3.1 Signal-to-Noise (S/N) ratio and main effects plot

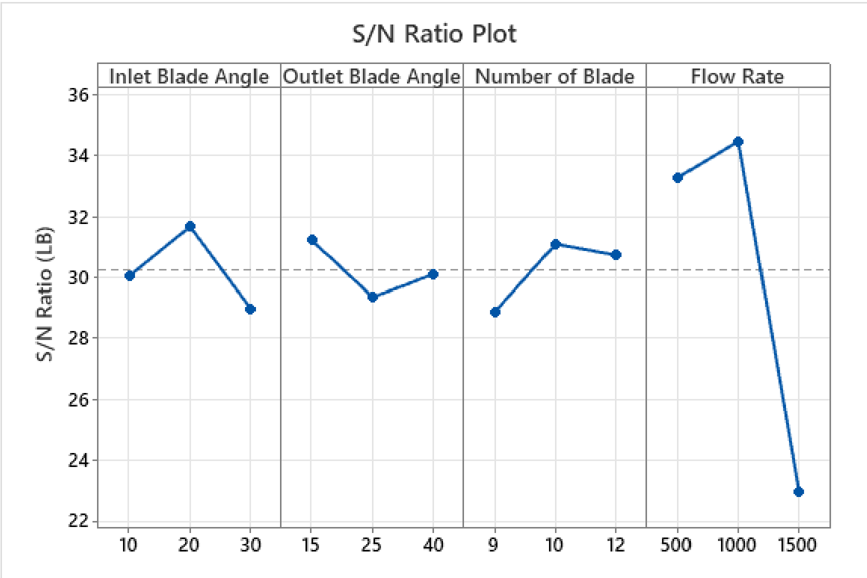
Table 4 presents the CFD simulation results based on the Taguchi L9 orthogonal array. Taguchi L9 is designed to evaluate the main effects of design parameters efficiently with a reduced number of simulations. Each simulation case was repeated three times to ensure consistency and reduce numerical variability. The signal-to-noise (S/N) ratio is used to evaluate the robustness of each parameter combination, with a higher S/N ratio, indicating greater robustness in efficiency performance.

Figure 7 show the signal-to-noise (S/N) ratio and the main effects plots for efficiency. The mean plot shows the average of efficiency, while the signal-to-noise ratio plot shows the robustness of efficiency with respect to the parameter variation [16], [29]. The highest S/N ratio is observed when the inlet blade angle is 20°, the outlet blade angle is 15°, the number of blades is 10, and the flow rate is 1000 m³/h. Among the investigated parameters, the flow rate is the most significant factor influencing the efficiency. The main effect plot shows a similar trend.

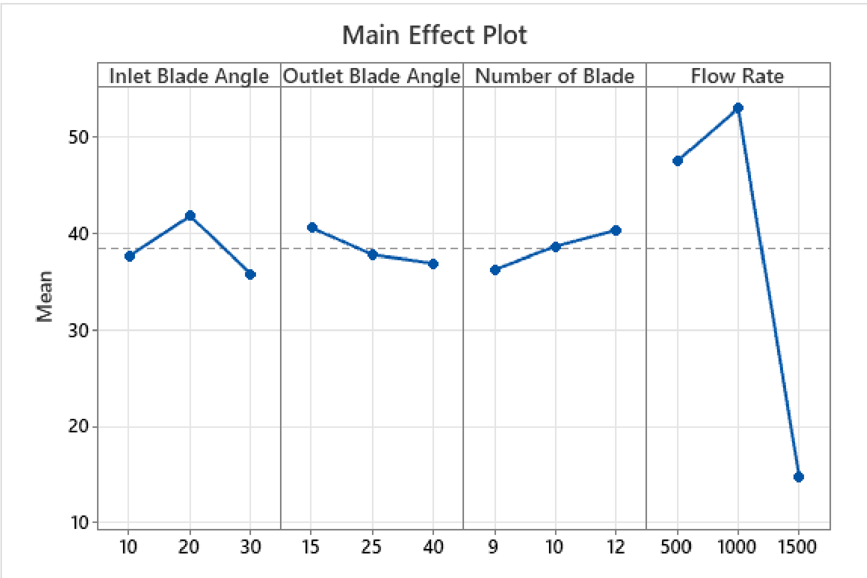
e-ISSN: 2621-8720 p-ISSN: 2621-9980

Table 4. Response table for S/N ratio and mean efficiency at each factor level

Exp.	Control Factor				Efficiency (%)				S/N Larger - The Better
	Inlet Blade Angle ($\beta 1$)	Outlet Blade Angle ($\beta 2$)	Number Of Blades (n)	Flow Rate (Q)	Result 1	Result 2	Result 3	Average	
1	10	15	9	1500	53.37	53.28	53.37	53.34	34.54
2	10	25	10	500	51.01	53.29	50.94	51.75	34.27
3	10	40	12	1000	14.45	14.42	14.34	14.40	23.17
4	20	15	10	1000	20.58	20.44	20.64	20.55	26.26
5	20	25	12	1500	53.56	49.37	53.44	52.13	34.32
6	20	40	9	500	52.45	52.05	53.40	52.63	34.42
7	30	15	12	500	55.16	54.28	53.71	54.38	34.71
8	30	25	9	1000	9.42	9.47	9.31	9.40	19.46
9	30	40	10	1500	46.15	41.92	42.72	43.60	32.77



(a)



(b)

Figure 7. (a) S/N ratio plot, and (b) Main effect plot

3.2 Analysis of variance (ANOVA)

ANOVA in the Taguchi method is used to identify the most influential factors, assess their statistical significance, and determine each factor's contribution percentage to the system response, as detailed in Table 5. ANOVA is widely used to enhance the results of Taguchi -CFD optimization [30], [31].

Table 5. Analysis of variance

Factor	DoF	SS	MS	F	p	Contribution (%)
Inlet angle	2	11.07	5.535	2.546	0.282	4.00
Outlet angle	2	9.70	4.85	2.23	0.31	3.50
Number of blades	2	4.35	2.173	1	0.5	1.57
Flow rate	2	251.83	125.92	57.92	0.017	90.93
Total	8	276.95				100

Significance level ($\alpha = 0.05$) is used. Analysis shows that the flow rate is the only statistically significant parameter ($p < 0.05$). The ANOVA results indicate that the flow rate (Q) is the most dominant factor (90.93%) influencing the centrifugal fan performance, while the inlet blade angle (β_1), outlet blade angle (β_2), and number of blades (n) show relatively low contributions of 4.00%, 3.50%, and 1.57%, respectively. This result is consistent with other studies, which report that flow rate significantly affects the overall performance of centrifugal fans. For example, [12], [32] reveal that flow rate variations significantly affect the distribution of velocity, pressure rise, and losses in the fan system.

The results also indicate that geometric parameters such as blade angles and blade number exhibit more subtle effects and are often associated with flow control mechanisms, such as reduced flow separation and improved pressure distribution, rather than dominating overall performance. Similar results were reported by [3], who found that geometry optimization improved efficiency but did not outweigh the influence of operating conditions. Therefore, the dominance of flow rate observed in this study is in good agreement with existing literature, while the relatively lower statistical contribution of geometric parameters does not diminish their physical importance in enhancing aerodynamic performance. The dominance of the flow rate is mainly due to the wide range of flow rates used in this study (500–1500 m^3/h). This causes significant changes in flow characteristics and loss mechanisms. As a consequence, the influence of geometric parameters becomes relatively less significant. A narrower range of flow rates might reduce this dominance and allow for a clearer assessment of geometric influences.

3.3 The efficiency comparison of optimal parameters with the reference

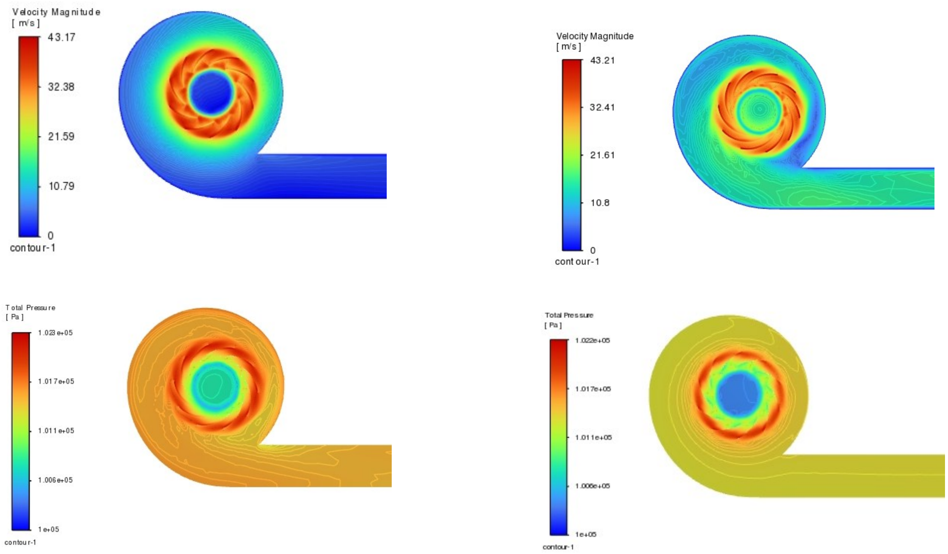
CFD simulations are performed to evaluate the efficiency improvement of the optimal configuration relative to the efficiency of the reference configuration. The optimal configuration is $\beta_1 = 20°$, $\beta_2 = 15°$, n = 10, and Q = 1000 m^3/h. The reference configuration is $\beta_1 = 30°$, $\beta_2 = 25°$, n = 10, and Q = 1000 m^3/h.

Table 6. Baseline confirmation test results

Run	Reference configuration (%)	Optimal configuration (%)
1	38.78	63.35
2	41.23	62.97
3	39.37	63.46

As shown in Table 6, the reference configuration exhibits efficiencies of 38.78%–41.23%, while the optimized configuration demonstrates higher efficiencies of 62.97%–63.46%. This indicates a significant improvement in performance under the same operating conditions. To ensure the reliability of the numerical results, the reference configuration has been validated against previously reported data [25], showing good agreement within an acceptable deviation. Based on this validation, the optimized configuration obtained from the Taguchi method demonstrates a consistent improvement in efficiency. These findings suggest that the proposed optimization approach is feasible in enhancing the efficiency of the centrifugal fan. The improvement in impeller efficiency obtained from the Taguchi optimization is consistent with established theoretical studies or previous research. The inlet blade angle (β_1) plays a critical role in minimizing the incidence angle between the incoming flow and the blade leading edge. A value of 20° provides better flow alignment, thereby reducing flow separation and associated losses at the impeller inlet. Similar observations have been reported by previous studies, where a properly set inlet blade angle significantly reduces incidence losses and improves flow attachment [32], [33]. The outlet blade angle (β_2) significantly influences the energy transfer and pressure rise within the impeller. A moderate outlet angle of 15° improves the conversion of mechanical energy into fluid energy, resulting in a more uniform pressure distribution and reduced wake formation. This is consistent with Euler's turbomachinery equation, where the tangential velocity component at the outlet contributes directly to the pressure rise. Previous studies have shown that smaller outlet blade angles can reduce flow separation and improve pressure recovery in centrifugal fans [34], [35].

The number of blades (n = 10) represents an optimal balance between flow guidance and blockage effects. A lower number of blades may lead to insufficient flow guidance, while a higher number increases friction and blockage, causing additional losses. The selected value ensures smoother flow passage and reduced turbulence intensity. Similar trends have been reported in the literature, where an optimal blade number minimizes aerodynamic losses and enhances efficiency [36], [37]. The operating flow rate (Q = 1000 m³/h) corresponds to a near-design condition where the fan operates close to its best efficiency point (BEP). Under these conditions, flow separation and recirculation are minimized, thereby improving aerodynamic performance. This behaviour is consistent with established fan performance theory and previous experimental and numerical studies [38].



(a) (b)

Figure 8. Velocity magnitude and total pressure distributions at the impeller for reference configuration (a), and optimal configuration (b)

To understand the reasons for this efficiency enhancement, Figure 8 plots the flow field contours in the impeller mid-plane section of both configurations. The reference configuration has a non-uniform velocity contour, with some regions of slow flow and even some signs of flow separation in the blade passages, all of which contribute to increased aerodynamic losses. The optimal configuration, however, has a more uniform velocity contour and a smoother acceleration in the flow through the impeller passages. The lack of slow-flow regions and separation signs suggests a better flow inside the centrifugal fan, resulting in lower losses. This is consistent with the Taguchi and ANOVA results, confirming that the efficiency enhancement is indeed due to a better flow distribution within the centrifugal fan.

4. Conclusion

In this study, a hybrid CFD and Taguchi method was employed to optimize the impeller geometry. The numerical investigation of the effect of the inlet blade angle, outlet blade angle, number of blades, and flow rate on efficiency is carried out. The optimal design parameters are determined as $\beta_1 = 20°$, $\beta_2 = 15°$, n = 10, and Q = 1000 m^3/h. Efficiency improves significantly by around 67%. The flow rate is the most important factor, followed by the inlet blade and outlet blade angles, and the number of blades. The ANOVA analysis was used to verify the results statistically. The analysis of the flow field revealed a more uniform flow velocity distribution and less flow separation for the optimized impeller. From the results, it is clear that the CFD-Taguchi framework can be an efficient tool for optimizing the centrifugal fan and can serve as a reference guide for its design. Future study will include experimental validation.

Author's declaration

Author contribution

Delima Yanti Sari: Conceptualization, Methodology, Supervision, Writing-Original Draft. **Bagas Santoso**: CFD Simulation, Literature review. **Hendri Nurdin**: Methodology review, Data Analysis and Interpretation, Project administration. **Hastuti**: Statistical analysis, Visualization. **Rifelino**: Technical review, Results Interpretation, Discussion Review, Visualization support. **Fitrah Qalbina**: Writing - Review, Proofreading. **Tsung-Liang Wu**: Writing-Review & Editing. **Dani Harmanto**: Writing-Review.

Funding statement

This study was financially supported by the Institute for Research and Community Service of Universitas Negeri Padang under research contract number 1763/UN35.15/LT/2024.

Data availability

The data obtained from the CFD simulations, which support the results of this study, are available from the corresponding author on reasonable request.

Acknowledgements

The authors gratefully acknowledge the Institute for Research and Community Service of Universitas Negeri Padang, the Laboratory of Machine Construction, and all who supported this study.

Competing interest

The authors declare that they have no competing financial interests or personal relationships that could influence the work reported in this study.

Ethical clearance

Not applicable

AI statement

The authors of this manuscript declare that artificial intelligence tools were not used to produce scientific content in this manuscript.

Publisher's and Journal's note

Universitas Negeri Padang as the publisher, and Editor of Teknomekanik state that there is no conflict of interest towards this article publication.

References

[1] V. K. Patel Babubhai, A. Chaudhari, A. Sharma, and V. Diwakar, "Experimental and numerical study on the influence of flow passages in centrifugal fan using computational fluid dynamics," *Engineering Research Express*, vol. 5, no. 2, p. 025030, Jun. 2023, https://doi.org/10.1088/2631-8695/accf00

[2] X. Zhao, J. Guan, T. Wang, X. Liu, Q. Xu, and J. Zhou, "Optimization of Impeller Structure Parameters of a Centrifugal Fan in a Powered Air-Purifying Respirator Power System," *Processes*, vol. 12, no. 2, p. 353, Feb. 2024, https://doi.org/10.3390/pr12020353

[3] P. Ragauskas, I. Tetsmann, and R. Jasevičius, "The Optimization of the Geometry of the Centrifugal Fan at Different Design Points," *Applied Sciences*, vol. 14, no. 8, p. 3530, Apr. 2024, https://doi.org/10.3390/app14083530

[4] R. A. Nabawi, E. Fadillah, H. S. Mulyadi, M. S. Fahrezi, and F. F. Putra, "A concave impeller: A new modified semi-open impeller for higher performance of centrifugal pumps," *Journal of Engineering Researcher and Lecturer*, vol. 4, no. 1, pp. 22–30, Apr. 2025, https://doi.org/10.58712/jerel.v4i1.178

[5] H. K. Sakran, M. S. Abdul Aziz, M. Z. Abdullah, C. Y. Khor, and M. R. R. Mohd Arif Zainol, "Impacts of impeller blade number on centrifugal pump performance under critical cavitation conditions," *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, vol. 142, Sep. 2024, https://doi.org/10.1177/09544089241278219

[6] X. Liu, Q. Dang, and G. Xi, "Performance Improvement of Centrifugal Fan by Using CFD," *Engineering Applications of Computational Fluid Mechanics*, vol. 2, no. 2, pp. 130–140, Jan. 2008, https://doi.org/10.1080/19942060.2008.11015216

[7] N. N. Bayomi, A. Abdel Hafiz, and A. M. Osman, "Effect of inlet straighteners on centrifugal fan performance," *Energy Convers. Manag.*, vol. 47, no. 18–19, pp. 3307–3318, Nov. 2006, https://doi.org/10.1016/j.enconman.2006.01.003

[8] G. R. H. Abo Elyamin, M. A. Bassily, K. Y. Khalil, and M. S. Gomaa, "Effect of impeller blades number on the performance of a centrifugal pump," *Alexandria Engineering Journal*, vol. 58, no. 1, pp. 39–48, Mar. 2019, https://doi.org/10.1016/j.aej.2019.02.004

[9] Z. Li, M. Zhu, Y. Liu, Z. Chen, and P. Luo, "Surrogate Model on the Extension Operation Range of an Isolated Centrifugal Fan," *Journal of Applied Fluid Mechanics*, vol. 17, no. 4, pp. 900–911, Apr. 2024, https://doi.org/10.47176/jafm.17.4.2148

[10] T. Liu, Y. Zhang, K. Li, Y. Wang, and J. Li, "Aerodynamic Optimization of a Centrifugal Fan Using Response Surface Methodology," *Open Journal of Fluid Dynamics*, vol. 09, no. 01, pp. 92–105, 2019, https://doi.org/10.4236/ojfd.2019.91007
[11] F. Meng, L. Wang, Y. Xiao, G. Xie, D. Zhang, and F. Zhao, "Multi-Objective Optimization for the Impeller Parameters of Centrifugal Fan Based on Kriging Model and GA-PSO Algorithm," *Recent Patents on Mechanical Engineering*, vol. 11, no. 3, pp. 242–250, Sep. 2018, https://doi.org/10.2174/2212797611666180622125804
[12] J. Zhang and M. Zangeneh, "Multi-Point, Multi-Objective Optimisation of Centrifugal Fans by 3D Inverse Design Method," *International Journal of Turbomachinery, Propulsion and Power*, vol. 8, no. 1, Mar. 2023, https://doi.org/10.3390/ijtpp8010008
[13] H. Ding, T. Chang, and F. Lin, "The influence of the blade outlet angle on the flow field and pressure pulsation in a centrifugal fan," *Processes*, vol. 8, no. 11, pp. 1–14, Nov. 2020, https://doi.org/10.3390/pr8111422
[14] Z. Hu, D. Wang, F. Gao, Y. Cao, and H. Wu, "Experimental investigation on cooling performance of vortex tube with rectifier using Taguchi method," *Case Studies in Thermal Engineering*, vol. 49, Sep. 2023, https://doi.org/10.1016/j.csite.2023.103373
[15] H. Fatahian, R. Mishra, F. F. Jackson, and E. Fatahian, "Design optimization of an innovative deflector with bleed jets to enhance the performance of dual Savonius turbines using CFD-Taguchi method," *Energy Convers. Manag.*, vol. 296, p. 117655, Nov. 2023, https://doi.org/10.1016/j.enconman.2023.117655
[16] K. L. Liaw *et al.*, "Optimization of a novel impulse gas turbine nozzle and blades design utilizing Taguchi method for micro-scale power generation," *Energy*, vol. 282, p. 129018, Nov. 2023, https://doi.org/10.1016/j.energy.2023.129018
[17] L. B. Kothe, A. P. Petry, T. D. J. Vecina, and J. L. R. Luz, "Optimization to increase energy efficiency of a sirocco centrifugal fan using computational fluid dynamics (CFD)," *Revista de Engenharia Térmica*, vol. 16, no. 1, p. 52, Jun. 2017, https://doi.org/10.5380/reterm.v16i1.62191
[18] F. Meng, L. Wang, W. Ming, and H. Zhang, "Aerodynamics Optimization of Multi-Blade Centrifugal Fan Based on Extreme Learning Machine Surrogate Model and Particle Swarm Optimization Algorithm," *Metals (Basel).*, vol. 13, no. 7, Jul. 2023, https://doi.org/10.3390/met13071222
[19] M. H. Shojaeefard and S. Saremian, "Analyzing the impact of blade geometrical parameters on energy recovery and efficiency of centrifugal pump as turbine installed in the pressure-reducing station," *Energy*, vol. 289, p. 130004, Feb. 2024, https://doi.org/10.1016/j.energy.2023.130004
[20] J. Fang, P. Yan, J. Qin, and H. Chen, "Research on Centrifugal Blower Design and Optimization Method Based on Mean Camber Line Optimization and CFD," *J. Phys. Conf. Ser.*, vol. 2419, no. 1, p. 012062, Jan. 2023, https://doi.org/10.1088/1742-6596/2419/1/012062
[21] H. Moon, J. Jeong, S. Park, K. Ha, and J. H. Jeong, "Numerical and experimental validation of vortex generator effect on power performance improvement in MW-class wind turbine blade," *Renew. Energy*, vol. 212, pp. 443–454, Aug. 2023, https://doi.org/10.1016/j.renene.2023.04.104
[22] R. P. Putra, D. Yuvenda, W. Afnison, R. Lapisa, Milana, and A. N. Fauza, "Design and crash test on a two-passenger city car frame using the finite element method," *Automotive Experiences*, vol. 7, no. 2, pp. 270–283, May 2024, https://doi.org/10.31603/ae.11306
[23] C. K. Huang and M. E. Hsieh, "Performance analysis and optimized design of Backward-Curved airfoil centrifugal blowers," *HVAC and R Research*, vol. 15, no. 3, pp. 461–488, 2009, https://doi.org/10.1080/10789669.2009.10390846
[24] R. N. Tiwari, C. A. Niccolini Marmont Du Haut Champ, F. Reggio, P. Silvestri, A. Traverso, and M. L. Ferrari, "Acoustic signature analysis of a bladeless blower," *Applied Acoustics*, vol. 208, Jun. 2023, https://doi.org/10.1016/j.apacoust.2023.109382

[25] S. Salmat, D. Yanti Sari, Y. Fernanda, and F. Prasetya, "SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan," *Journal of Engineering Researcher and Lecturer*, vol. 2, no. 3, pp. 94–103, Nov. 2023, https://doi.org/10.58712/jerel.v2i3.104

[26] A. Rais, Y. Fernanda, A. K, and A. Kurniawan, "Analysis of Fluid Flow in Pipeline and Cyclone Systems for Preduster Coal Mill Indarung V Factory Using Flow Simulation," *MOTIVECTION : Journal of Mechanical, Electrical and Industrial Engineering*, vol. 5, no. 2, pp. 303–316, Apr. 2023, https://doi.org/10.46574/motivection.v5i2.224

[27] N. S. Rajneesh, C. H. A. Kumar, M. Sharma, and S. Pavuluri, "CFD Analysis Flow Through Cryogenic Expansion Turbine," in *AIP Conference Proceedings*, American Institute of Physics Inc., Nov. 2023. https://doi.org/10.1063/5.0158518

[28] B. D. Baloni, Y. Pathak, and S. A. Channiwala, "Centrifugal blower volute optimization based on Taguchi method," *Comput. Fluids*, vol. 112, pp. 72–78, May 2015, https://doi.org/10.1016/j.compfluid.2015.02.007

[29] K. Qaissi, O. Elsayed, M. Faqir, and E. Essadiqi, "Aerodynamic Optimization of Trailing-Edge-Serrations for a Wind Turbine Blade Using Taguchi Modified Additive Model," *Energies (Basel).*, vol. 16, no. 3, Feb. 2023, https://doi.org/10.3390/en16031099

[30] J. P. Agrawal, N. Somani, and N. K. Gupta, "Optimizing electric discharge machining parameters using Taguchi method and ANOVA: A study on AISI D2 steel machining with copper electrodes and materials science progress," *Next Materials*, vol. 10, Jan. 2026, https://doi.org/10.1016/j.nxmate.2025.101361

[31] Y. E. Cetin, B. E. Yuce, M. Kriegel, P. Wargocki, and P. V. Nielsen, "Optimization of classroom window design using CFD and the Taguchi method," *Journal of Building Engineering*, vol. 106, Jul. 2025, https://doi.org/10.1016/j.jobe.2025.112580

[32] Z. Yu, S. Li, W. He, W. Wang, D. Huang, and Z. Zhu, "Numerical Simulation of Flow Field for a Whole Centrifugal Fan and Analysis of the Effects of Blade Inlet Angle and Impeller Gap," *HVAC&R Res.*, vol. 11, no. 2, pp. 263–283, Apr. 2005, https://doi.org/10.1080/10789669.2005.10391137

[33] F.-N. Meng, L.-W. Wang, G.-Z. Xie, F. Zhao, D.-H. Zhang, and W.-L. Du, "Effects of Blade Inlet Angle on Flow Field of Centrifugal Fan," *DEStech Transactions on Engineering and Technology Research*, no. icmeca, Jul. 2017, https://doi.org/10.12783/dtetr/icmeca2017/11924

[34] M. Bai, Z. Liu, Y. Ling, and H. Tan, "Effect of impeller structure on aerodynamic performance and noise reduction of a small multi-blade centrifugal fan," *Sci. Technol. Built Environ.*, vol. 30, no. 6, pp. 563–578, Jul. 2024, https://doi.org/10.1080/23744731.2024.2357526

[35] D. Ravi and T. K. R. Rajagopal, "Numerical Investigation on the Effect of Geometric Shape and Outlet Angle of a Bladeless Fan for Flow Optimization using CFD Techniques," *International Journal of Thermofluids*, vol. 15, Aug. 2022, https://doi.org/10.1016/j.ijft.2022.100174

[36] S. Zhou, K. Yang, W. Zhang, K. Zhang, C. Wang, and W. Jin, "Optimization of Multi-Blade Centrifugal Fan Blade Design for Ventilation and Air-Conditioning System Based on Disturbance CST Function," *Applied Sciences*, vol. 11, no. 17, p. 7784, Aug. 2021, https://doi.org/10.3390/app11177784

[37] S. Varun Ch, K. Anantharaman, and G. Rajasekaran, "Effect of blade number on the performance of centrifugal fan," in *Materials Today: Proceedings*, Elsevier Ltd, Jan. 2023, pp. 1143–1152. https://doi.org/10.1016/j.matpr.2022.09.185

[38] J. Dixon and C. Hall, *Fluid Mechanics and Thermodynamics of Turbomachinery*. Elsevier, 2010. https://doi.org/10.1016/C2009-0-20205-4