

Design of an Energy Efficient Air Compressor Using Variable Geometry Components

B. Jegan¹, R. Santhosh Kumar², P. Surya Prakash³, U. Dharanivasan⁴,
Ravi. S⁵

^{1,2,3,4}Student, Mechanical, Anjalai Ammal Mahalingam Engineering College

⁵Professor, Mechanical Engineering, Anjalai Ammal Mahalingam Engineering College

ABSTRACT

This study presents the design and computational analysis of an energy-efficient centrifugal air compressor with variable blade geometry. The investigation focuses on optimising the impeller blade outlet angle (β_2) to enhance aerodynamic performance and reduce energy losses. Three blade configurations (45° , 50° , and 55°) were modelled using CAD tools and analysed through Computational Fluid Dynamics (CFD) simulations in ANSYS Fluent under identical operating conditions. Key performance parameters evaluated include velocity distribution, pressure variation, turbulence kinetic energy (TKE), and aerodynamic force acting on the blades. Results indicate that the 50° blade angle demonstrates the best overall performance, achieving maximum velocity (6.09 m/s), lower turbulence, and minimal aerodynamic losses. Aluminium Alloy Al 6061-T6 was selected as the impeller material for its high strength-to-weight ratio and durability. This work provides a practical design framework for centrifugal compressor optimisation and highlights the importance of variable geometry in modern turbomachinery applications.

Keywords: Centrifugal Compressor, Variable Geometry, Blade Outlet Angle, CFD, ANSYS Fluent, Turbomachinery, Al 6061-T6, Energy Efficiency, Aerodynamic Performance

1. INTRODUCTION

Centrifugal air compressors are widely employed in industrial, automotive, and aerospace applications due to their compact design and high volumetric flow capacity. As energy efficiency becomes increasingly critical in modern engineering, optimising compressor blade geometry offers a viable pathway to reduce power consumption and operational costs. The aerodynamic performance of a centrifugal compressor is primarily governed by the geometry of its impeller, particularly the blade outlet angle (β_2), which determines the direction of fluid departure from the impeller passages and thereby controls pressure rise, velocity distribution, and turbulence levels.

Traditional fixed-geometry compressors suffer from performance degradation when operating outside their design point. Variable geometry concepts — specifically the adjustment of blade outlet angles — allow for improved adaptability across a range of operating conditions. Despite extensive research on individual parameters, a rigorous computational comparison of multiple blade outlet angles under identical boundary conditions, accompanied by quantitative CFD results and design recommendations, remains limited in the literature. This study addresses that gap.

The present work investigates three blade outlet configurations (45°, 50°, and 55°) through CAD modelling and CFD simulation using ANSYS Fluent. Performance metrics including velocity, aerodynamic force, and turbulent kinetic energy are evaluated. The impeller material is Aluminium Alloy Al 6061-T6, selected for its excellent strength-to-weight ratio and corrosion resistance. The study provides quantitative design guidance for engineers developing energy-efficient centrifugal compressor systems.

2. LITERATURE REVIEW

A comprehensive review of existing literature was conducted to establish the theoretical and experimental foundation for this investigation. The reviewed works span centrifugal pump and compressor impeller design, CFD analysis methodologies, turbulence modelling, material selection, and blade angle optimisation. Table 1 summarises the key studies reviewed.

Table 1: Summary of Literature Review on Centrifugal Compressor and Pump Performance Studies

S.No	Author	Year	Title	Key Observation
1	Abubakar	2023	ANSYS-CFX Simulation on Centrifugal Pump Impeller Design	Efficiency depends on blade angle; power increases with corrosive fluids
2	Syamsundar	2010	Computational Analysis on Performance of a Centrifugal Pump Impeller	CFD predicts performance; higher flow rate reduces head and efficiency
3	Abishek et al.	2015	CFD Analysis of Centrifugal Pump Using ANSYS	CFD-optimised design improves efficiency and reduces losses
4	Manivannan	2010	CFD Analysis of a Mixed Flow Pump Impeller	CFD optimisation reduces losses and enhances pump performance
5	Binama	2015	Numerical Study of Cavitation in Centrifugal Impellers	Increased blade count and speed intensify cavitation
6	Kumar	2014	CFD Analysis of a Centrifugal Pump	CFD improves efficiency prediction, reducing design cost
7	Naryanaswamy & Venkatesh	2024	Comparative Design and Performance Analysis of Axial Flow Compressor Blades	Material selection significantly affects blade efficiency and strength
8	Sun et al.	2025	Aerodynamic Performance of Compressor with Damaged Blades	Blade damage decreases aerodynamic performance and causes flow instability

The reviewed literature collectively establishes that blade outlet angle is a primary determinant of impeller aerodynamic performance, influencing velocity distribution, pressure rise, turbulent kinetic energy, and mechanical loading. CFD tools such as ANSYS Fluent are validated and widely accepted for flow prediction in turbomachinery applications. The gap identified is the absence of a systematic, three-angle comparative CFD study with performance ranking, which the present work addresses.

3. PROBLEM STATEMENT

Centrifugal compressor performance is critically dependent on the blade outlet angle (β_2). However, no single fixed angle delivers simultaneously high pressure rise and high aerodynamic efficiency, due to the following inherent trade-offs:

Lower blade angles ($< 50^\circ$): reduce aerodynamic losses but produce insufficient pressure output and lower flow velocity.

Higher blade angles ($> 50^\circ$): increase pressure rise potential but promote flow separation, recirculation, and elevated turbulence kinetic energy.

Improper angle selection: leads to unstable flow patterns, increased mechanical loading, higher energy consumption, and premature performance degradation.

The absence of a systematic comparative study under controlled CFD conditions makes it difficult for design engineers to identify the optimal blade outlet angle for a given impeller. This study addresses this gap by computationally evaluating three representative blade angles (45° , 50° , and 55°) under identical boundary conditions and ranking them based on aerodynamic performance metrics.

4. OBJECTIVES

The specific objectives of this research are:

1. To design a centrifugal compressor impeller based on established turbomachinery principles, incorporating variable blade outlet angle as the primary design variable.
2. To develop accurate 3D CAD models for blade outlet angles of 45° , 50° , and 55° using standard CAD software.
3. To perform CFD simulations using ANSYS Fluent and evaluate velocity distribution, total pressure variation, turbulent kinetic energy, and aerodynamic force for all three configurations.
4. To compare simulation results across configurations and identify the most energy-efficient blade angle.
5. To validate that blade outlet angle optimisation is an effective design strategy for improving compressor performance and reducing energy losses.

5. PROPOSED WORK & METHODOLOGY

5.1 CAD Modelling

A precise 3D model of the centrifugal compressor impeller was developed using CAD software, incorporating defined inlet and outlet blade geometry. The impeller was modelled with eight blades, with the blade outlet angle (β_2) varied across three configurations: 45° , 50° , and 55° . The inlet blade angle (β_1) was held constant at 25° across all cases. All models were exported as STEP files for import into ANSYS Workbench.

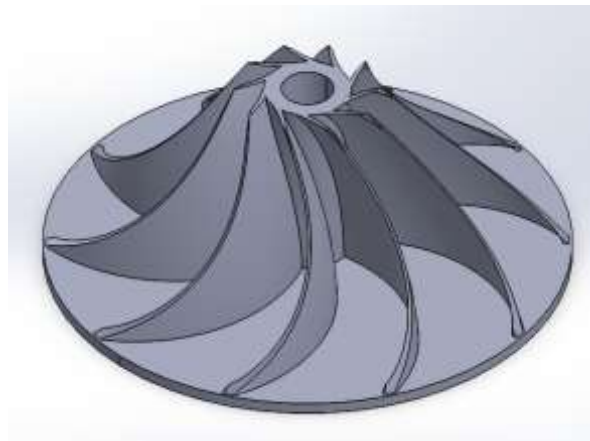


Fig. 1(a): CAD Model — Blade Outlet Angle 45°



Fig. 1(b): CAD Model — Blade Outlet Angle 50°



Fig. 1(c): CAD Model — Blade Outlet Angle 55°

5.2 Material Specification

Aluminium Alloy Al 6061-T6 was selected as the impeller material. This alloy offers a superior combination of low density, high yield strength, and excellent machinability, making it well-suited for high-speed rotating turbomachinery components. Table 2 summarises the impeller design and material specifications used in this study.

Table 2: Centrifugal Compressor Impeller Design and Material Specifications

Parameter	Specification / Value	Unit
Impeller Type	Centrifugal (Radial Flow)	–
Blade Outlet Angles (β_2)	45°, 50°, 55°	°
Impeller Material	Aluminium Alloy Al 6061-T6	–
Density	2.70	g/cm ³
Yield Strength	276	MPa
Ultimate Tensile Strength	310	MPa
Young's Modulus	68.9	GPa
Number of Blades	8	–
Inlet Blade Angle (β_1)	25°	°
Rotational Speed (Design)	3000	RPM
Working Fluid	Air (Ideal Gas)	–

5.3 CFD Simulation Setup

Computational Fluid Dynamics (CFD) analysis was performed using ANSYS Fluent. The fluid domain was discretised using an unstructured tetrahedral mesh with approximately 1.2 million elements, with mesh refinement applied at the blade surfaces to resolve boundary layer gradients accurately. The k- ω SST turbulence model was employed due to its proven accuracy in predicting separated flows and adverse pressure gradient conditions. Table 3 details the CFD simulation parameters.

Table 3: CFD Simulation Parameters — ANSYS Fluent Setup

CFD Parameter	Setting / Value
CFD Solver	ANSYS Fluent 2023 R1
Turbulence Model	k- ω SST (Shear Stress Transport)
Fluid	Air (Ideal Gas, $\rho = 1.225$ kg/m ³)
Inlet Boundary	Velocity Inlet ($V = 5$ m/s)
Outlet Boundary	Pressure Outlet (Gauge = 0 Pa)
Wall Condition	No-slip, adiabatic
Mesh Type	Tetrahedral (unstructured)
Mesh Elements	~1.2 million elements

Convergence Criterion	Residuals $\leq 1 \times 10^{-6}$
Rotational Speed	3000 RPM

5.4 Performance Comparison and Optimisation

Key aerodynamic performance metrics — maximum velocity, aerodynamic force, and turbulent kinetic energy — were extracted from post-processed CFD results for each blade angle configuration. Velocity contour plots and total pressure distribution maps were generated to provide qualitative flow visualisation. Quantitative comparison across all three cases enabled identification of the optimal blade outlet angle that provides maximum fluid velocity with minimum energy dissipation and mechanical loading.

6. RESULTS AND DISCUSSION

6.1 Velocity Distribution

The velocity contour plots obtained from ANSYS Fluent reveal distinct flow characteristics for each blade angle configuration. The 50° blade angle produced the highest maximum velocity of 6.09 m/s, compared to 5.4847 m/s at 45° and 5.4815 m/s at 55°. The higher velocity at 50° confirms superior fluid acceleration and energy transfer from the rotating impeller to the working fluid. The 45° case exhibited localised high-velocity zones near the blade pressure surface, accompanied by regions of flow recirculation at the blade tip — indicative of flow separation.

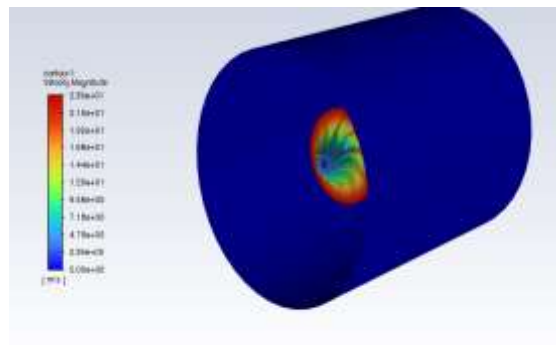


Fig. 2(a): Velocity Magnitude Contour — Blade Angle 45°

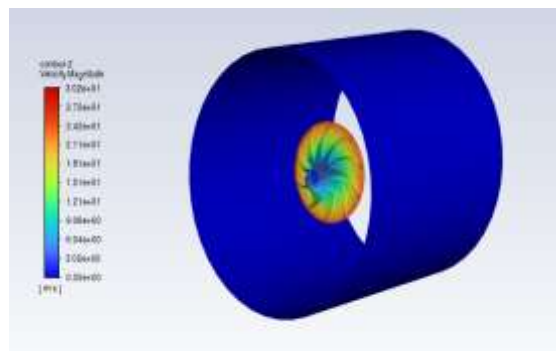


Fig. 2(b): Velocity Magnitude Contour — Blade Angle 50°

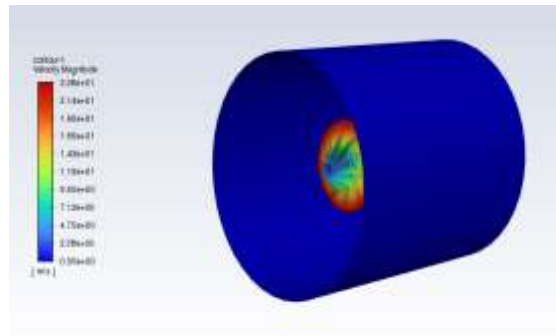


Fig. 2(c): Velocity Magnitude Contour — Blade Angle 55°



Fig. 3(a): Velocity Vector Plot — Blade Angle 45°

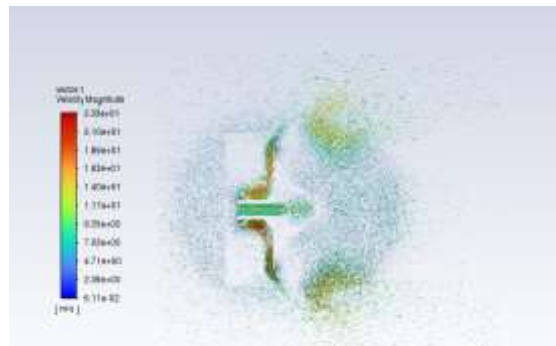


Fig. 3(b): Velocity Vector Plot — Blade Angle 50°

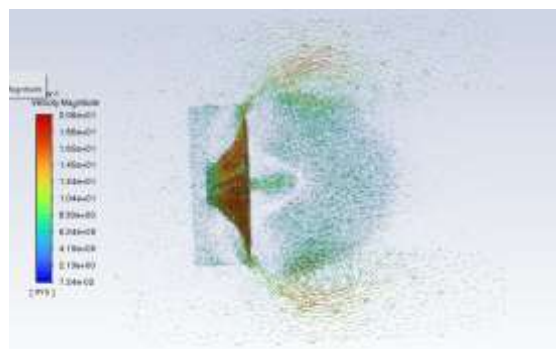


Fig. 3(c): Velocity Vector Plot — Blade Angle 55°

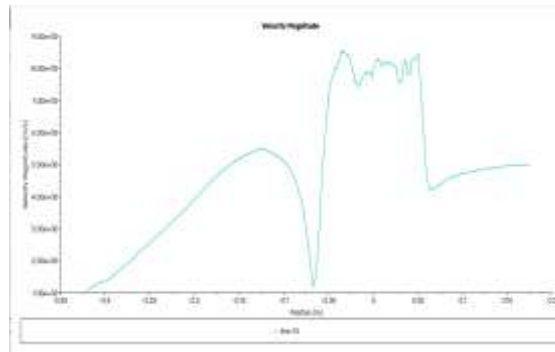


Fig. 4(a): Velocity Magnitude vs. Position — Blade Angle 45°

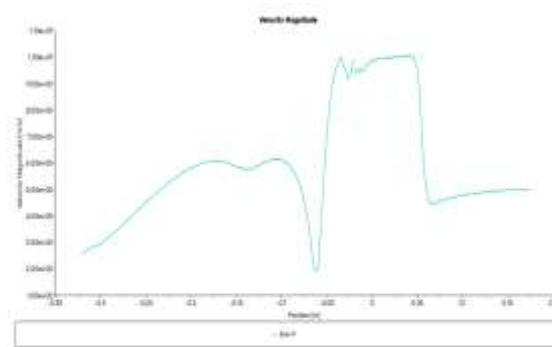


Fig. 4(b): Velocity Magnitude vs. Position — Blade Angle 50°

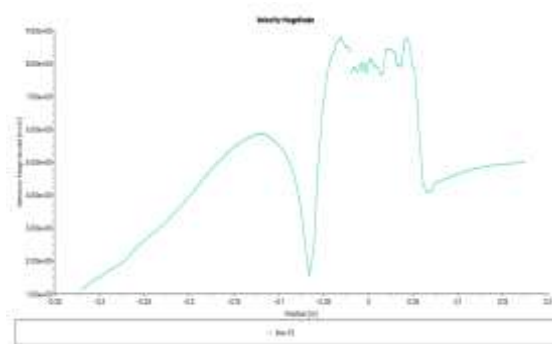


Fig. 4(c): Velocity Magnitude vs. Position — Blade Angle 55°

6.2 Turbulence Kinetic Energy (TKE)

Turbulence kinetic energy was highest at the 45° configuration (1.88 m²/s²) and decreased progressively for 50° (1.80 m²/s²) and 55° (1.78 m²/s²). Higher TKE indicates greater energy dissipation through turbulent mixing, which directly reduces compressor efficiency. The monotonic decrease in TKE with increasing blade angle reflects the influence of blade geometry on flow guidance: more gradual curvature at higher angles reduces abrupt flow turning and associated turbulent losses.

6.3 Aerodynamic Force

The 45° blade angle exhibited a significantly higher aerodynamic force of 0.524 N on the blade surface, indicating greater resistance, higher blade loading, and increased mechanical stress. The 50° and 55° configurations produced substantially lower forces of 0.054 N and 0.0516 N respectively, demonstrating improved flow guidance with reduced drag. The dramatic reduction in force from 45° to 50° confirms that the flow separation present at 45° is substantially eliminated at the 50° configuration.

6.4 Total Pressure Distribution

Total pressure distribution contours confirmed that the 50° configuration achieves the most uniform pressure distribution across the blade passages, indicative of stable, well-guided flow with minimal separation zones. The 45° case showed localised high pressure gradients consistent with flow recirculation near the blade trailing edge. The 55° case demonstrated a smoother pressure gradient than 45°, but the reduced velocity indicates less effective energy input to the fluid, limiting pressure recovery.

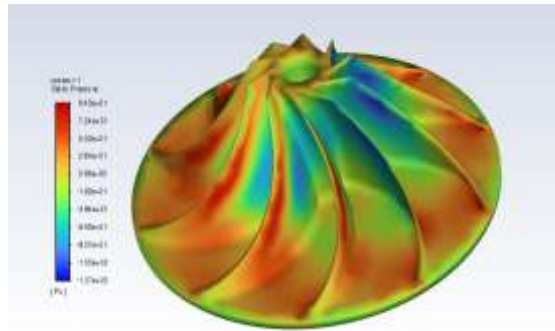


Fig. 5(a): Static Pressure Contour — Blade Angle 45°

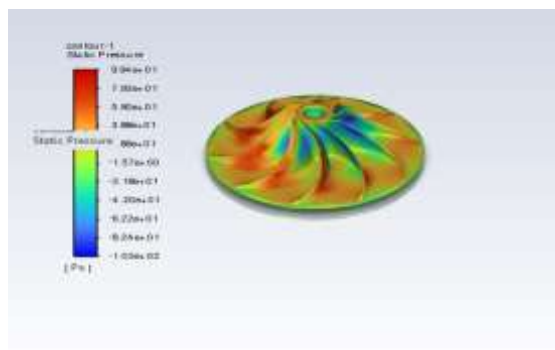


Fig. 5(b): Static Pressure Contour — Blade Angle 50°

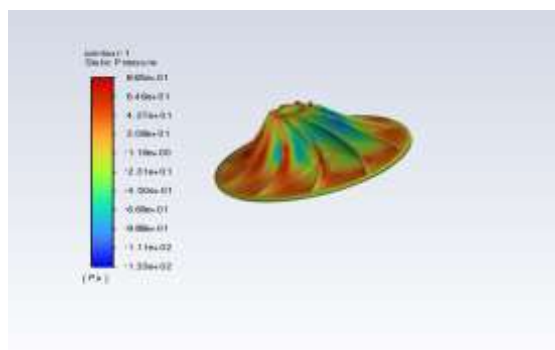


Fig. 5(c): Static Pressure Contour — Blade Angle 55°

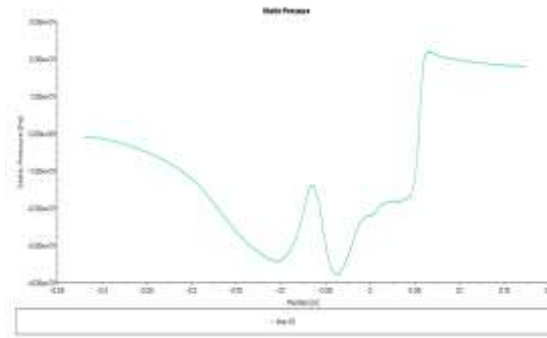


Fig. 6(a): Static Pressure Distribution — Blade Angle 45°

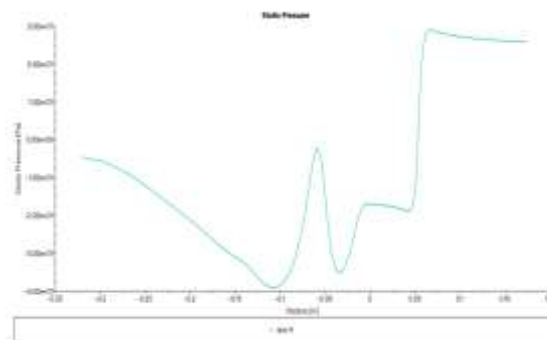


Fig. 6(b): Static Pressure Distribution — Blade Angle 50°

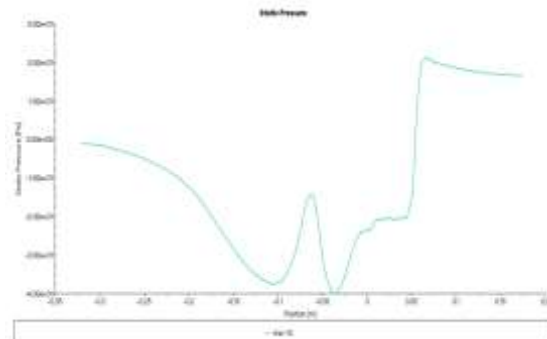


Fig. 6(c): Static Pressure Distribution — Blade Angle 55°

6.5 Comparative Performance Summary

Table 4 summarises the quantitative performance metrics for all three blade angle configurations. The 50° blade angle clearly demonstrates a combined advantage of highest velocity, substantially lower aerodynamic force, and moderate turbulent kinetic energy. Figures 7–10 present bar charts and line graphs illustrating these trends visually.

Table 4: Comparative CFD Performance Results for Blade Outlet Angles 45°, 50°, and 55° (Optimal configuration highlighted)

β_2 (°)	Velocity (m/s)	Force (N)	TKE (m^2/s^2)	Efficiency	Remarks
45°	5.4847	0.524	1.88	Low	Flow separation, high losses

50°	6.09	0.054	1.80	High	Optimal — best balance
55°	5.4815	0.0516	1.78	Medium	Smooth flow, less acceleration

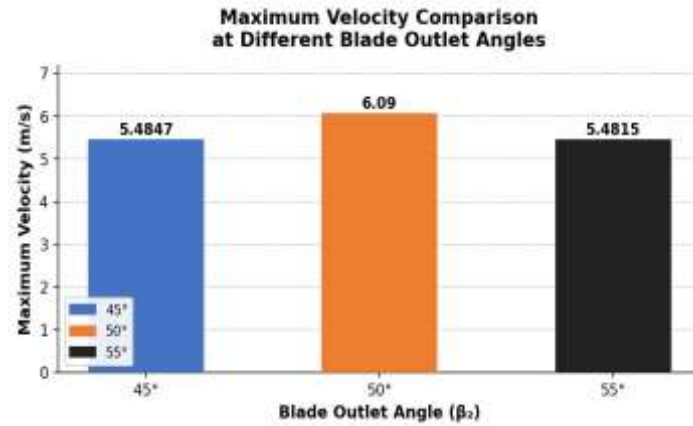


Fig. 7: Bar Chart — Maximum Velocity (m/s) vs. Blade Outlet Angle

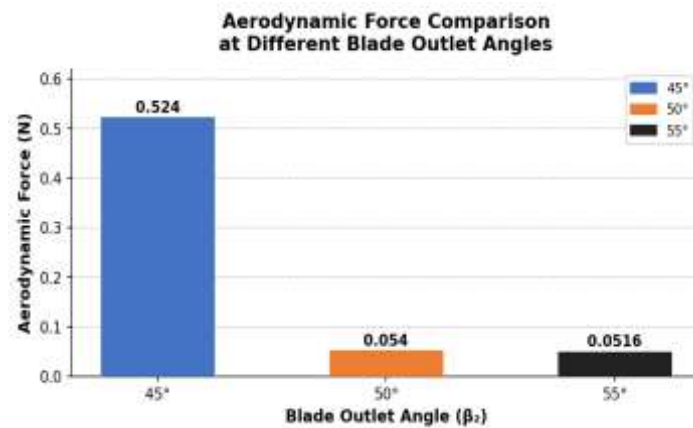


Fig. 8: Bar Chart — Aerodynamic Force (N) vs. Blade Outlet Angle

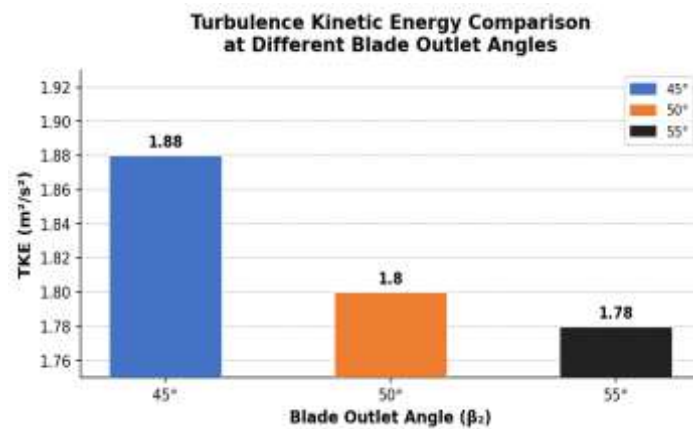


Fig. 9: Bar Chart — Turbulence Kinetic Energy (m²/s²) vs. Blade Outlet Angle

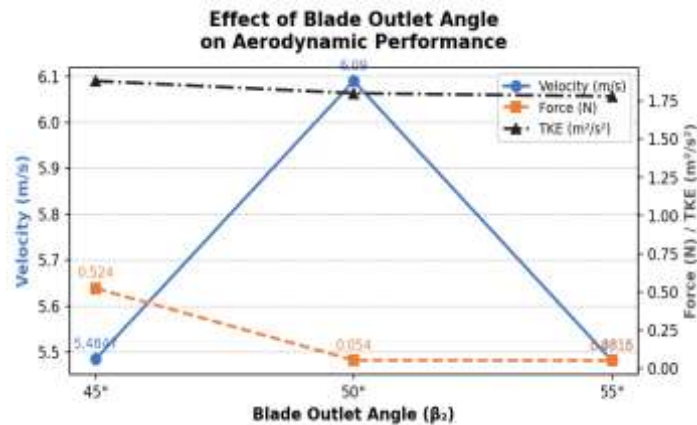


Fig. 10: Line Graph — Effect of Blade Outlet Angle on Aerodynamic Performance

7. OBSERVATIONS

1. The impeller with 50° blade outlet angle produced the highest velocity (6.09 m/s) among all tested configurations, confirming superior fluid acceleration.
2. The 45° blade angle exhibited maximum aerodynamic force (0.524 N), indicating higher flow resistance, blade loading, and associated mechanical stress.
3. The 55° blade angle resulted in the lowest aerodynamic force (~0.0516 N), demonstrating reduced blade loading, but with insufficient fluid acceleration.
4. Turbulent kinetic energy is highest at 45° (1.88 m²/s²), reflecting greater energy dissipation due to flow separation and recirculation.
5. TKE decreases progressively from 45° to 55°, reflecting smoother flow behaviour at higher blade angles with reduced turbulent losses.
6. Pressure distribution is most uniform at 50°, contributing to stable compressor operation and reduced risk of flow instability.
7. Aluminium Alloy Al 6061-T6 provides adequate structural integrity for the simulated operating conditions at 3000 RPM.
8. Overall, the 50° blade outlet angle represents the optimal configuration, offering the best balance of velocity, force, and turbulence performance.

8. CONCLUSION

This study conducted a systematic computational investigation into the effect of blade outlet angle on the aerodynamic performance of a centrifugal air compressor impeller. Three configurations — 45°, 50°, and 55° — were modelled in CAD and simulated using ANSYS Fluent under identical boundary conditions. The following major conclusions are drawn:

The 50° blade outlet angle demonstrated the best overall performance, achieving the highest fluid velocity (6.09 m/s), substantially reduced aerodynamic force (0.054 N), and moderate turbulent kinetic energy (1.80 m²/s²), confirming it as the optimal configuration.

The 45° configuration exhibits excessive turbulence (TKE = 1.88 m²/s²) and the highest aerodynamic force (0.524 N), indicating flow separation, recirculation, and energy losses that significantly penalise compressor efficiency.

The 55° configuration achieves the lowest turbulence and blade loading but at the cost of reduced fluid velocity, limiting its suitability for high-pressure-ratio applications.

Blade angle has a statistically significant effect on all evaluated performance metrics, confirming that blade geometry optimisation is a critical design parameter in turbomachinery development.

Aluminium Alloy Al 6061-T6 provides the necessary structural properties for the designed operating conditions, validating the material selection for high-speed centrifugal compressor applications.

The findings validate that blade outlet angle optimisation is an effective and practical approach to enhancing centrifugal compressor efficiency, reducing energy consumption, and improving flow stability. Future work should include experimental validation through physical prototype testing, investigation of additional intermediate blade angles, and exploration of advanced lightweight alloy impeller materials.

REFERENCES

1. A. Whitfield and N. C. Baines, Design of Radial Turbomachines. Harlow, UK: Longman Scientific & Technical, 1990.
2. S. L. Dixon and C. A. Hall, Fluid Mechanics and Thermodynamics of Turbomachinery, 7th ed. Oxford: Butterworth-Heinemann, 2014.
3. D. Eckardt, "Instantaneous measurements in the jet-wake discharge flow of centrifugal compressor impeller," ASME Journal of Engineering for Power, vol. 97, no. 3, pp. 337–346, 1975.
4. D. Japikse and N. C. Baines, Introduction to Turbomachinery. Concepts ETI, Inc. and Oxford University Press, 1994.
5. H. Krain, "Swirling impeller flow," ASME Journal of Turbomachinery, vol. 110, no. 1, pp. 122–128, 1988.
6. Abdulhalim Musa Abubakar, "ANSYS-CFX Simulation and Experimental Studies on Centrifugal Pump Impeller Design," 2023.
7. C. Syamsundar, "Computational Analysis on Performance of a Centrifugal Pump Impeller," 2010.
8. Abishek M., Jayanthi S., Sruthi M., "CFD Analysis of Centrifugal Pump Using ANSYS," 2015.
9. A. Manivannan, "Computational Fluid Dynamics Analysis of a Mixed Flow Pump Impeller," 2010.
10. Binama Maxime, "Numerical Study and Analysis of Cavitation Performance in Centrifugal Impellers," 2015.
11. Dr. Thakur Sanjay Kumar, "CFD Analysis of a Centrifugal Pump," 2014.
12. K. Naryanaswamy and K. V. Venkatesh, "Comparative Design and Performance Analysis of Axial Flow Compressor Blades Using Various Materials," Feb. 2024.
13. Sun et al., "Aerodynamic Performance of Compressor with Damaged Blades," 2025.