

Article

Aerodynamic Optimisation of a Tandem Blade Centrifugal Compressor Through Parametric Analysis of Blade Angles and Count

Mustafa Ertürk Söylemez  and Salih Özer * 

Department of Mechanical Engineering, Muş Alparslan University, Muş 49100, Türkiye;
e.soylez@alparslan.edu.tr

* Correspondence: s.oz@alparslan.edu.tr

Abstract

This study advances the performance of a tandem-blade centrifugal compressor through a parametric Computational Fluid Dynamics (CFD) methodology integrated with Response Surface Methodology (RSM). Numerical simulations were executed by solving steady-state Reynolds-Averaged Navier–Stokes (RANS) equations utilising the Shear Stress Transport (SST) $k-\omega$ turbulence model on a validated structured hexahedral mesh. Local sensitivity analysis identified the hub outlet angle and hub inlet angle as the primary geometric parameters affecting pressure ratio and isentropic efficiency, respectively. Flow-field visualisations confirmed that the tandem configuration effectively re-energises the boundary layer, thereby reducing separation and enhancing pressure recovery. Using a Multi-Objective Genetic Algorithm (MOGA), an optimal blade design comprising 22 blades was determined, achieving a maximum isentropic efficiency of 95.23% and a total pressure ratio of 1.416. These findings provide valuable quantitative insights for the optimal design of tandem impellers and highlight the effectiveness of integrating CFD-based sensitivity analysis with multi-objective optimisation techniques.

Keywords: centrifugal compressor; tandem blade; CFD optimisation; response surface methodology; MOGA

1. Introduction

Centrifugal compressors are extensively employed owing to their capability to attain high pressure ratios within a compact configuration across applications such as industrial, aerospace, and energy sectors. However, the occurrence of flow instabilities—including stall and surge—frequently constrains the operational range and efficiency of these machines. Consequently, recent years have witnessed comprehensive investigations aimed at enhancing compressor efficiency and operational stability, focusing on advanced blade geometries, diffuser improvements, and numerical optimisation techniques. These studies have demonstrated substantial efficiency gains when implemented in industry and related fields. For instance, Shaban, optimised the design of bladeless diffusers, achieving approximately a 3.8% increase in the pressure coefficient and a reduction in total losses [1]. Similarly, Demir et al. [2] provided evidence through detailed CFD analysis on industrial fans that their efficiencies could be improved. Such research underscores the persistent necessity for ongoing efforts to improve the aerodynamic performance of centrifugal compressors.



Academic Editors: Juan C. García Castrejón and Laura Lilia Castro Gomez

Received: 6 January 2026

Revised: 1 February 2026

Accepted: 2 February 2026

Published: 4 February 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

A prominent approach to performance enhancement is the adoption of tandem (dual) blade configurations. In a tandem setup, two blade rows—namely, inducer and exducer—are positioned consecutively within the same blade passage, facilitating distributed work input across the flow. This design strategy enhances boundary-layer control, permits higher loading within a single stage, and reduces the number of stages required to achieve a given pressure increase. The literature indicates that tandem blade geometries can effectively regulate aerodynamic load distribution and enhance stage stability. For example, Hagelstein et al., observed that compressors equipped with tandem wheels achieved higher overall pressure ratios and experienced lower losses compared to single-row configurations, albeit with a narrowed operational range [3]. Galloway et al. [4] emphasize that the pressure ratio, efficiency, and operating range are fundamental performance criteria in centrifugal compressors; while vaned diffusers provide high efficiency, they have a narrow operating range. It is noted that aerodynamic instabilities such as stall and surge, which occur at low flow rates, limit compressor stability. It is demonstrated that a new body improvement design developed for the vaned diffuser significantly expands the stable operating range at low flow rates and high-pressure ratios, as shown by experimental and CFD analyses. Moreover, reducing the front blade (inducer) thickness by 20% significantly improved efficiency relative to baseline designs. Cheng et al., demonstrated that the circumferential positioning (clocking) of tandem blade rows could alter efficiency by several percentage points [5]. Conversely, experimental research by Josuhn-Kadner [6] indicated that although tandem configurations significantly influence flow structures, their impact on total stage efficiency remains limited. These findings suggest that the performance effects of tandem blade geometries are highly sensitive to specific geometric parameters and blade angles.

In addition to tandem configurations, fundamental geometric variables—such as blade number and blade angle distribution—directly affect the flow behavior and efficiency of compressors. An increase in blade count generally enhances flow guidance and elevates pressure rise; however, excessive blade numbers may lead to channel blockage and uneven load sharing, resulting in efficiency losses. Previous sensitivity analyses have identified blade count as a critical parameter in the performance of centrifugal compressors. Additionally, the inlet blade angle modulates flow attachment and attack angle, while the outlet angle (or backsweep) influences diffusion characteristics and kinetic energy distribution at the exit. A larger backsweep angle typically reduces flow separation and improves efficiency, though excessively high values can compromise system stability and induce stall conditions. Denton, while examining losses in blade surfaces, tip leaks, and end wall regions in turbomachinery, emphasizes that many types of losses are still not fully understood and that estimates are largely based on empirical correlations, and states that physical improvements that can be developed could lead to efficiency gains [7]. Jin et al. [8] propose a prediction model and a multi-objective optimization algorithm for multi-bladed centrifugal fans and state that they applied these to the optimization design of a multi-bladed centrifugal fan. They demonstrate that the aerodynamic and noise performance of the optimized fan has improved, indicating that this provides a reference for the optimized design of such fans.

Recent advancements in CFD methods have considerably improved the analysis and optimization capabilities concerning geometric effects in centrifugal compressors. High-fidelity solvers such as ANSYS CFX facilitate detailed examinations of flow fields and enable precise calculation of performance metrics, including isentropic efficiency and total pressure ratio. CFD-based optimization frequently incorporates genetic algorithms, Design of Experiments (DOE), and surrogate modeling techniques. For example, Söylemez et al., and

Ziliang et al., demonstrate that the aerodynamics of a tandem centrifugal impeller can be optimized and improved using genetic algorithms and numerical analyses [9,10]. Likewise, Gua et al. [11] employed a Kriging-based multi-objective optimization methodology to concurrently refine multiple blade parameters, yielding significant efficiency improvements. When combined with modern optimization algorithms, CFD fosters systematic exploration of the trade-off between efficiency and pressure ratio across the design space.

An examination of existing literature reveals a notable gap in systematic studies analyzing the impact of blade angle parameters specifically within tandem blade centrifugal compressors. Prior research has primarily concentrated on structural parameters such as blade number, thickness, or spacing. For example, optimized blade count and thickness but did not explore the influence of blade angle variations (inlet and outlet angles) on compressor performance. This study aims to investigate parametrically the effects of blade angles and blade number in a tandem blade centrifugal compressor via CFD simulations utilizing ANSYS CFX. Various combinations of inducer and exducer blade angles are tested, with the resultant flow fields, pressure ratios, and efficiencies being systematically compared and evaluated [11–14].

The overarching objective of this research is to incorporate blade angles as independent variables within an optimisation framework for tandem blade centrifugal compressors. Unlike previous investigations that predominantly focus on blade number and thickness, this study provides a quantitative assessment of how blade angles influence flow patterns, pressure ratios, and isentropic efficiencies. While prior research has largely addressed structural factors such as blade count and thickness, this study fills a notable gap by treating the 3D blade metal angle distributions at both hub and shroud as independent continuous variables within a MOGA framework. To identify the optimal combination of blade angle and blade number, a data-driven optimization methodology integrating RSM with a MOGA is established. This approach aims to extend beyond qualitative judgments, offering practical and systematic guidelines for the aerodynamic design of tandem wheel configurations.

2. Methodology

To examine the influence of design parameters, including the number of blades, hub inlet angle, hub outlet angle, shroud inlet angle, shroud outlet angle, and flow rate, on the performance of a radial compressor with tandem blades, an impeller with an existing performance map documented in the literature was chosen. The specific design parameters of this propeller were taken from the article published [15].

Although the centrifugal compressor was aerodynamically designed for a nominal rotational speed of 23,000 rpm, the numerical simulations were conducted at a reduced rotational speed of 17,473 rpm. This choice was dictated by the experimental constraints of the test rig, where the rotor of the high-speed electric motor coupled to the compressor shaft was limited to a maximum rotational speed of 17,473 rpm. To ensure a consistent and meaningful comparison between numerical predictions and experimental measurements, all CFD simulations for both baseline validation and optimisation studies were therefore performed at this rotational speed. Consequently, the numerical analysis reflects the actual operating conditions of the experimental setup rather than the nominal design speed.

A comprehensive analysis was conducted on studies focused on streamlining the design of radial compressors, leading to the selection of an appropriate design methodology. Employing this methodology, the compressor can be parametrically modeled to meet specified requirements. The detailed design process is illustrated in Figure 1.

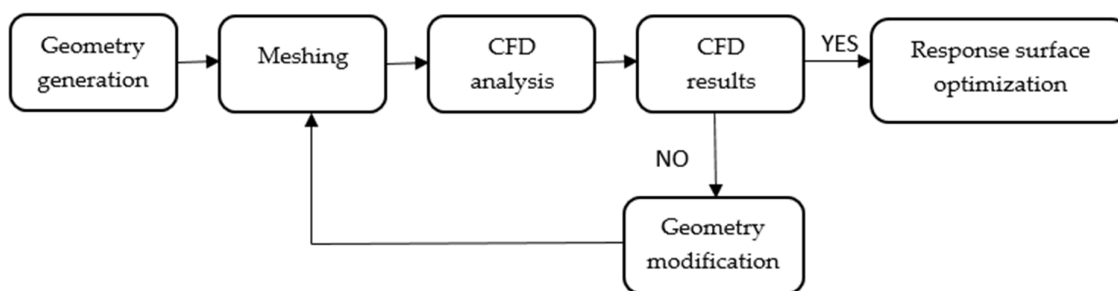


Figure 1. Design methodology [9].

2.1. Physical Model and Governing Equations

The aerodynamic flow field within the tandem blade centrifugal compressor is modelled through the resolution of the three-dimensional, steady-state, compressible RANS equations. The conservation principles pertaining to mass, momentum, and energy are addressed within a stationary reference frame as outlined below [16]:

1. Continuity Equation: The conservation of mass for a steady flow is defined as:

$$\nabla \cdot (\rho \mathbf{U}) = 0 \quad (1)$$

where ρ is the fluid density and $\mathbf{U}$ is the velocity vector.

2. Momentum Equation: The conservation of momentum is expressed as:

$$\nabla \cdot (\rho \mathbf{U} \otimes \mathbf{U}) = -\nabla p + \nabla \tau + S_M \quad (2)$$

Here, p denotes the pressure, and S_M represents the external body forces. The viscous stress tensor, τ , relates to the strain rate for a Newtonian fluid as follows:

$$\tau = \mu (\nabla \mathbf{U} + (\nabla \mathbf{U})^T) - \frac{2}{3} \delta \nabla \cdot \mathbf{U} \quad (3)$$

where μ is the dynamic viscosity and δ is the identity matrix.

3. Energy Equation: To account for high-speed effects and heat transfer, the total energy equation is utilized:

$$\nabla \cdot (\rho \mathbf{U} h_{tot}) = \nabla \cdot (\lambda \nabla T + \mathbf{U} \cdot \tau) + S_E \quad (4)$$

where h_{tot} is the total enthalpy, related to the static enthalpy h by $h_{tot} = h + \frac{1}{2} \mathbf{U}^2$. The term λ represents the thermal conductivity and S_E denotes energy sources.

4. Equation of State: The working fluid is modeled as an ideal gas, specifically air, to account for compressibility effects. The relationship among pressure, density, and temperature is described by the following equation:

$$p = \rho R T \quad (5)$$

where R is the specific gas constant.

Turbulence Modeling: The Shear Stress Transport (SST) k - ω model is utilised to close the RANS equations. This model is favoured in centrifugal compressor investigations due to its reliable performance in predicting flow separation under adverse pressure gradients. It employs the k - ω formulation within the inner boundary layer regions and transitions to a k - ϵ formulation in the free stream [17].

The transport equations governing turbulent kinetic energy (k) and the specific dissipation rate (ω) are:

$$\nabla \cdot (\rho U k) = \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \beta^* \rho k \omega \quad (6)$$

$$\nabla \cdot (\rho U \omega) = \nabla \left[\left(\mu + \frac{\mu_t}{\sigma_\omega} \right) \nabla \omega \right] + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + (1 - F_1) 2\rho \frac{1}{\sigma_{\omega 2} \omega} \nabla k \nabla \omega \quad (7)$$

The blending function F_1 ensures the transition between the models. Numerical convergence is achieved when the root-mean-square (RMS) residuals of the governing equations fall below the target of 10^{-4} .

2.2. Geometrical Model and Parametrization

The fundamental geometry of the tandem blade centrifugal impeller was developed utilising ANSYS BladeGen, a tool specifically tailored for turbomachinery applications. The impeller features a tandem blade configuration to optimise aerodynamic load distribution and enhance boundary layer management. In this configuration, an inducer and an exducer blade are incorporated within a single blade row, facilitating re-energisation of the flow between blade elements and minimising the risk of flow separation.

For systematic optimization, the impeller geometry was established using a parametric framework based on the meridional profile and blade angle (β) distributions. The blade geometry was parametrised at two principal spanwise locations, namely the hub and the shroud. The variation in blade angles along the normalized meridional coordinate (M') was selected as the primary geometric control mechanism, given its direct influence on flow turning, blade loading, and pressure increase within the impeller passage.

2.2.1. Parameter Definition

The geometric design space of the tandem impeller was delineated utilising a set of key parameters identified through preliminary sensitivity analyses. These parameters include the leading-edge (LE) and trailing-edge (TE) blade angles at both the hub and shroud sections, as well as the primary blade count. Collectively, these variables afford sufficient flexibility to regulate the aerodynamic behavior of the impeller while preserving geometric integrity.

The hub inlet angle ($\beta_{\text{hub,in}}$) and hub outlet angle ($\beta_{\text{hub,out}}$) predominantly influence the incidence characteristics and flow turning at the blade root. In contrast, the shroud inlet angle ($\beta_{\text{shr,in}}$) and shroud outlet angle ($\beta_{\text{shr,out}}$) impact tip flow dynamics, including secondary flows and potential leakage-related losses. The primary blade count (Z) was incorporated as a discrete design variable due to its significant influence on flow guidance, work input, and frictional losses. All parameters were defined within physically plausible bounds to guarantee manufacturability and numerical stability throughout the optimization process.

2.2.2. Blade Angle Distributions

The blade angle distributions along the meridional direction for both the hub and shroud are depicted schematically in Figures 2 and 3. These distributions delineate the gradual variation in the blade metal angle (β) from the leading edge to the trailing edge and represent the blade twist implemented to accommodate three-dimensional flow effects.

At the hub section, the blade angle distribution was specifically designed to regulate blade loading in the vicinity of the root region, where the flow exhibits heightened sensitivity to incidence and separation attributable to pronounced curvature and centrifugal effects.

The parameters $\beta_{\text{hub,in}}$ and $\beta_{\text{hub,out}}$ establish the boundary conditions of the distribution and influence the overall curvature of the blade at the hub.

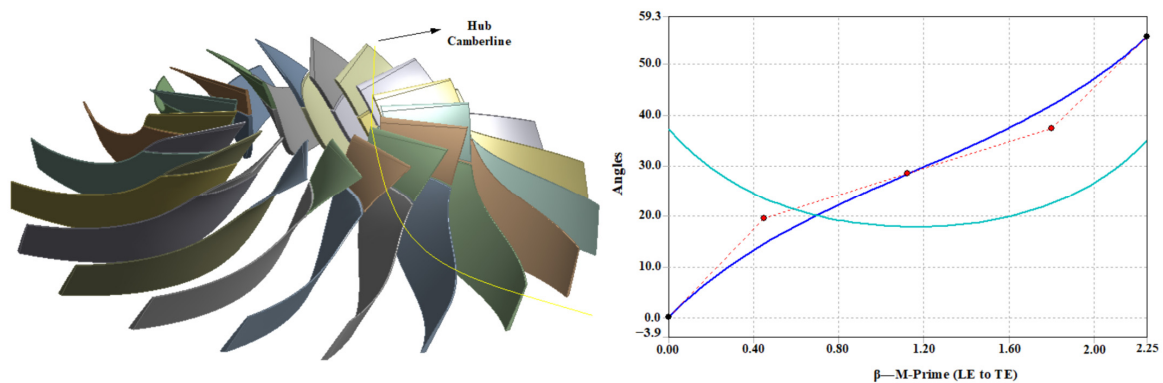


Figure 2. Hub blade metal angle (β) distribution along the normalized meridional coordinate (M') and the corresponding three-dimensional tandem blade geometry.

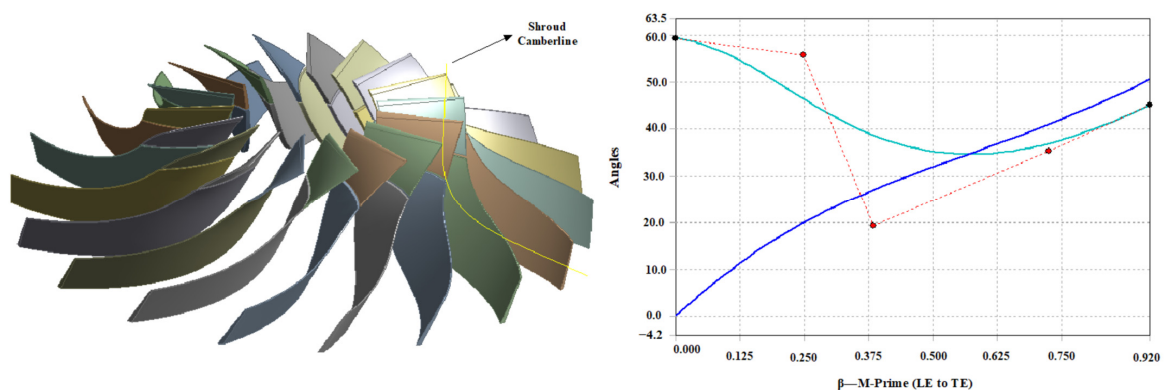


Figure 3. Shroud blade metal angle (β) distribution along the normalized meridional coordinate (M') and the corresponding three-dimensional tandem blade geometry.

The inlet and outlet hub angles delineate the blade curvature at the root section and regulate local blade loading and flow turning characteristics. Likewise, the shroud blade angle distribution was engineered to manage the flow behavior proximate to the blade tip. The shroud inlet and outlet angles are pivotal in mitigating adverse pressure gradients, controlling secondary flow development, and stabilizing the flow structure within the blade passage. The coordinated adjustment of hub and shroud angle distributions facilitates effective three-dimensional flow regulation across the entire impeller span.

The shroud inlet and outlet angles control the flow behavior near the blade tip, influencing secondary flow development and overall aerodynamic stability.

2.2.3. Computational Domain Simplification

Following the geometric parameterization, the finalized three-dimensional impeller model was transferred to ANSYS DesignModeler for fluid domain extraction. To reduce computational cost while preserving numerical accuracy for steady-state simulations, a single periodic blade passage was selected as the computational domain. Rotational periodic boundary conditions were applied to the pitch-wise surfaces, allowing the single-passage model to represent the full 360° impeller while significantly reducing the number of computational cells. This approach is widely adopted in turbomachinery CFD studies and provides reliable predictions of aerodynamic performance under steady operating conditions. The resulting computational domain includes the complete tandem blade geometry, hub, and shroud surfaces required to accurately resolve the internal flow field.

While this simplification provides high computational efficiency for optimization, it should be noted that the periodic single-passage model neglects potential annulus non-uniformities and circumferential interactions between tandem rows, such as clocking effects, which are acknowledged as influential in the literature.

2.3. Meshing Strategy and Grid Independence

The computational domain was discretized employing a high-grade structured hexahedral mesh developed in ANSYS TurboGrid, a tool specially optimized for turbomachinery blade passages. The utilisation of hexahedral elements guarantees enhanced numerical stability and convergence relative to tetrahedral grids, especially in accurately representing the intricate flow physics within high-speed centrifugal compressors.

2.3.1. Meshing Strategy

As illustrated in Figure 4 the mesh topology follows an automated ATM3D (Automated Topology and Meshing) approach to ensure high orthogonality and low aspect ratios in the critical regions of the tandem blades. To accurately resolve the boundary layer and capture the near-wall velocity gradients, 5 inflation layers were meticulously applied to the blade surfaces, hub, and shroud. The first layer thickness was carefully calculated to achieve a target of $y^+ < 5$ across the entire domain. Verification of the converged numerical solutions confirmed that the average y^+ value was maintained at approximately 3.2 on the blade surfaces. This resolution ensures full compatibility with the SST $k-\omega$ turbulence model's requirements for high-fidelity near-wall treatment.

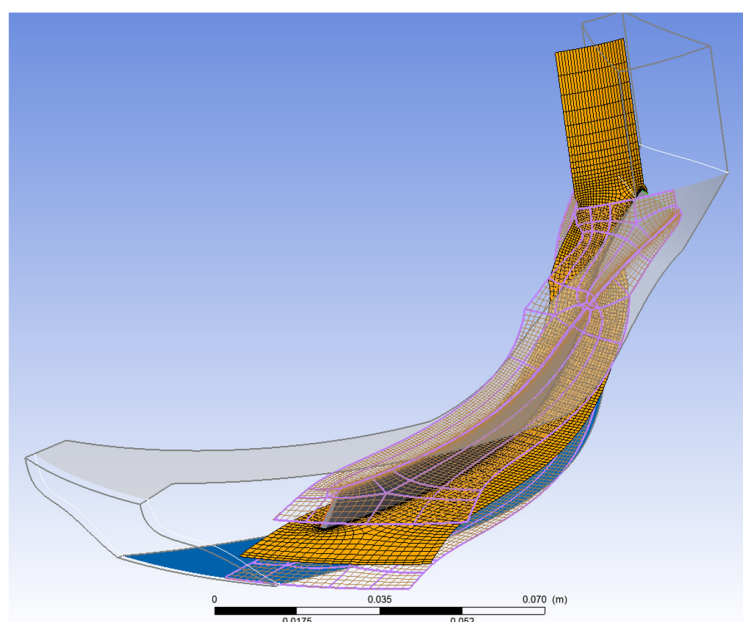


Figure 4. Structured hexahedral mesh of the periodic tandem blade passage. The inlet and outlet planes used for the evaluation of the total-to-total isentropic efficiency and pressure ratio are also indicated.

2.3.2. Grid Independence Study

To guarantee that the numerical predictions are independent of grid resolution and to reduce discretization errors, a systematic grid independence study was conducted. The total-to-total pressure ratio was chosen as the principal performance indicator and assessed across seven distinct structured hexahedral mesh configurations, with the total number of elements ranging from approximately 50,000 to 1,325,000.

Figure 5 delineates the variation in the total-to-total pressure ratio as a function of mesh density. As observed, significant fluctuations are evident in the estimates of the total

pressure ratio for meshes with fewer than 400,000 elements. This behaviour indicates that discretisation errors become dominant due to pronounced pressure gradients, secondary flow structures, and potential separation and re-attachment regions within tandem blade passages, which cannot be accurately captured with coarser meshes. Such oscillatory convergence behaviours are particularly sensitive to mesh resolution in internal turbomachinery flows. The literature emphasises that grid convergence should be systematically reported using multiple metrics, rather than relying solely on a single metric [18]. When the network attains 412,272 elements, a distinct convergence trend in the pressure ratio becomes apparent; subsequent refinements (up to approximately 1.32 million elements) show that the estimated variation in the total pressure ratio diminishes to a negligible level. The fact that the difference between the mesh with 412,272 elements and the finest mesh remains below 0.05% indicates that the solution has reached mesh independence with respect to this metric. Although the Grid Convergence Index (GCI) methodology is widely recommended for formal uncertainty quantification in CFD verification studies, the present work does not explicitly compute GCI values. Instead, mesh independence is demonstrated through systematic mesh refinement and the observation of asymptotic convergence behaviour of the total-to-total pressure ratio. The negligible variation observed beyond 412,272 elements indicates that discretisation errors have a minimal influence on the reported performance metrics. This approach is considered sufficient for the comparative design optimisation framework adopted in this study, where relative performance trends between design configurations are of primary interest. In conclusion, considering that a large number of design points need to be solved during the optimisation process, the mesh with 412,272 elements was selected as the baseline mesh for all stable RANS analyses in terms of the computational cost–accuracy balance. In the mesh independence study, it was observed that the pressure ratio difference between the network with 412,272 elements and the finest mesh ($\sim 1.32 \times 10^6$) remained below 0.05%. This difference and the mesh refinement ratio r , which is equal to the big letter N sub 1 divided by big letter N sub 2, raised to the power of one-third, equals 1.476, were used to perform a GCI-like upper bound calculation. This indicates that the numerical uncertainty originating from the mesh for PR is approximately $\pm 0.05\%$. The prediction errors of the surrogate (RSM) models are reported with cross-validation RMSE values: 0.002958 (approximately $\pm 0.23\%$ at $PR \approx 1.306$) for PR and 0.007539 (approximately $\pm 0.79\%$ at $\eta \approx 0.952$). This choice is generally consistent with the literature on turbomachinery CFD studies, which typically report solution meshes on the order of $\sim 10^6$ elements for performance metrics (e.g., total pressure ratio, isentropic efficiency). However, here it is demonstrated that similar reliability can be achieved with a lower number of elements due to the very small change in the target metric [19].

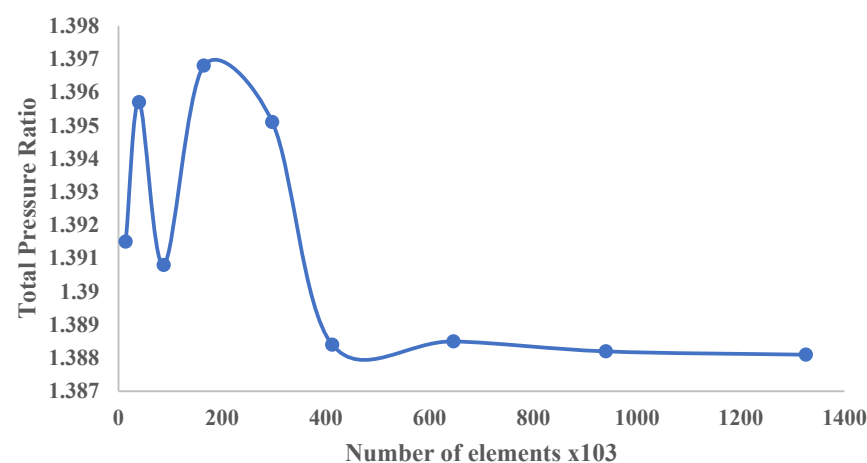


Figure 5. Mesh independence study.

2.4. Numerical Setup and Boundary Conditions

The numerical simulations were conducted utilising the ANSYS CFX solver. The computational domain comprised a singular blade passage with rotational periodicity interfaces designated on the lateral surfaces. The impeller domain was configured to operate at a rotational speed of 17,473 rpm. The boundary conditions were meticulously established to emulate the generation of the compressor performance map:

- **Inlet:** A stationary frame was applied with a Total Pressure of 101,325 Pa and a Total Temperature of 288.15 K. Turbulence intensity was set at a moderate level (5%) and a turbulence viscosity ratio of 10, following standard practices for centrifugal compressor CFD validation studies.
- **Outlet:** To regulate the operating point and facilitate the mapping of the compressor, a mass flow rate boundary condition was employed as the output parameter.
- **Walls:** All solid surfaces, including the hub, shroud, and blades, were modelled as adiabatic, no-slip walls.
- **Interfaces:** Periodic boundary conditions were imposed on the passage sides to simulate the complete annulus effect.

2.5. Sensitivity Analysis and Response Surface Validation Method

In order to quantify the influence of each design variable on the aerodynamic performance of the tandem blade centrifugal compressor, a local sensitivity analysis was performed utilising the Response Surface Optimization module within ANSYS Workbench. The response surface models were developed based on a series of CFD-generated design points, wherein key geometric and operational parameters were systematically varied within predefined bounds. The design space was sampled using an optimal space-filling DOE strategy implemented in ANSYS Workbench to ensure a uniform and representative coverage of the parameter space.

The optimization was conducted under a set of geometric and operational constraints to ensure physically meaningful, manufacturable, and aerodynamically stable designs. The hub and shroud blade inlet and outlet angles, blade count, and operating mass flow rate were constrained within specified ranges defined in this study. These constraint ranges were selected to allow sufficient design flexibility while maintaining stable compressor operation and avoiding near-surge and choking conditions. The complete set of design variables, together with their corresponding bounds and variable types, is summarized in Table 1.

Table 1. Design variables and constraint ranges used in the MOGA optimization.

Name	Lower Bound	Upper Bound	Variable Type
$\beta_{\text{hub,in}}$ (degree)	−5	5	Continuous
$\beta_{\text{hub,out}}$ (degree)	50	60	Continuous
$\beta_{\text{shr,in}}$ (degree)	55	65	Continuous
$\beta_{\text{shr,out}}$ (degree)	40	50	Continuous
Blade counts	14-16-18-20-22	-	Discrete
Mass flow rates	0.4	0.8	Continuous

While the sensitivity analysis formally evaluates the influence of both geometric parameters and the mass flow rate, the primary purpose of this analysis is to rank the geometric design variables. The mass flow rate is included as a bounded operating parameter to provide a physically realistic operating context and to assess robustness, and is not treated as a geometric optimisation variable or used to guide the subsequent optimisation process.

To guarantee the reliability of the optimisation and sensitivity results, the accuracy of the constructed response surface models was quantitatively evaluated using the Goodness of Fit analysis in ANSYS Workbench. The coefficient of determination (R^2) was employed to assess the goodness of fit between the CFD results and the surrogate model predictions, while the Root Mean Square Error (RMSE) was used to quantify the prediction error across the design space. The accuracy metrics obtained for each response variable, based on both the learning points and cross-validation, are summarised in Table 2.

Table 2. Accuracy metrics of the response surface models for the pressure ratio and isentropic efficiency.

Response Variable	R^2	RMSE	Cross-Validation R^2	Cross-Validation RMSE
Total to total ratio	0.99767	0.0005367	0.92913	0.0029581
Isentropic efficiency	0.99514	0.0012204	0.91462	0.0075387

3. Results and Discussion

3.1. Performance Comparison and Validation

In this section, the performance of the optimized design obtained through the numerical framework is compared with the initial experimental results to evaluate the effectiveness of the optimization and the validity of the flow physics. It should be noted that while the nominal design specifications of the compressor impeller are defined as a pressure ratio of 1.68 at 23,000 rpm, the experimental validation was conducted at a reduced rotational speed of 17,473 rpm, yielding a baseline total pressure ratio of 1.306. This specific operating condition was adopted to ensure a consistent comparison between the numerical results and the available experimental data. As explained in Section 2, experimental measurements were conducted at a reduced rotational speed of 17,473 rpm due to the operational limitations of the electric motor, and the same speed was adopted in the validation CFD simulations. The total pressure ratio is analyzed across a range of mass flow rates as the primary performance metric.

In the present study, the isentropic efficiency refers to the rotor-only total-to-total isentropic efficiency, evaluated between the impeller inlet and impeller exit planes. The inlet plane is located immediately upstream of the impeller leading edge, while the outlet plane is defined at the impeller trailing edge. No diffuser or return channel is included in the computational domain. All flow quantities are mass-flow-averaged over the corresponding planes. The efficiency is computed using the total-to-total definition as given by:

$$\eta_{tt} = \frac{\left(\frac{P_{t,out}}{P_{t,in}}\right)^{\frac{\gamma-1}{\gamma}} - 1}{\frac{T_{t,out}}{T_{t,in}} - 1}$$

The numerical simulations are conducted on a single-passage rotor domain with adiabatic wall boundary conditions. Therefore, the reported isentropic efficiency values represent upper-bound numerical estimates and are intended for relative performance comparison between design configurations rather than direct stage-level efficiency prediction.

The experimental test configuration includes a vaneless diffuser downstream of the impeller, whereas the numerical domain is limited to a single-passage, rotor-only model. In the CFD simulations, surface roughness, tip clearance, disk friction, and leakage flows are not explicitly modelled, and hydraulically smooth, adiabatic wall conditions are assumed. The experimental total pressure measurements are obtained at stations located upstream of the impeller inlet and downstream of the diffuser exit. These modelling and hardware

differences partially explain the systematically higher performance levels predicted by the CFD simulations compared to the experimental results.

Figure 6 illustrates the comparison between the experimental data and the results of the optimized design analysis. The numerical results for the optimum configuration follow the same characteristic trend as the experimental data, where the total pressure ratio decreases as the mass flow rate increases. The CFD predictions overestimate the experimental pressure ratio by approximately 4–8%, which is within the error margin commonly reported in centrifugal compressor validation studies. Such systematic offsets are generally attributed to idealised CFD assumptions, including the neglect of surface roughness, tip and leakage flows, and diffuser losses, as well as simplified boundary conditions [7,14,20].

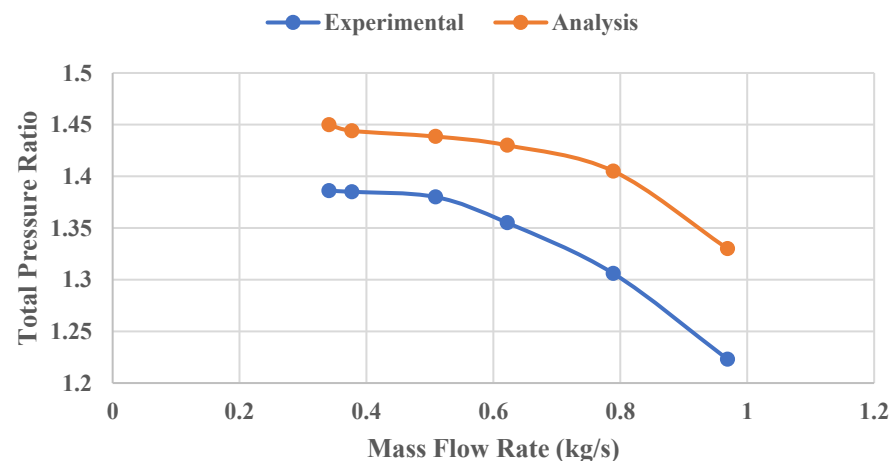


Figure 6. Comparison of total pressure ratio versus mass flow rate between experimental data and optimized design analysis.

It should be noted that the present validation focuses on the total-to-total pressure ratio, which is the primary performance parameter available from the experimental measurements conducted at the reduced rotational speed of 17,473 rpm. The isentropic efficiency values reported in this study are therefore purely numerical predictions derived from CFD simulations and are not directly validated against experimental efficiency data. Consequently, absolute efficiency levels should be interpreted as relative performance indicators for comparing different design configurations within the same numerical framework rather than as direct representations of industrial performance.

Previous studies have shown that the inclusion of surface roughness, tip clearance, disk friction, and diffuser losses typically reduces the predicted isentropic efficiency by approximately 5–10 percentage points and the total pressure ratio by 3–6%, depending on operating conditions [21,22]. Accordingly, the numerical efficiency values reported in the present study should be regarded as upper-bound estimates. Experimental isentropic efficiency data are not available from the test rig; therefore, quantitative CFD experiment efficiency offsets for efficiency cannot be provided. Nevertheless, the observed agreement in pressure ratio trends indicates that the dominant aerodynamic mechanisms are captured reliably, and the optimised tandem blade geometry provides consistent numerical performance improvements across the investigated operating range. The systematic overestimation of efficiency and pressure ratio is partly due to the neglect of tip clearance, disk friction, and annulus non-uniformities. However, the primary objective of this study is to evaluate relative aerodynamic gains. Although the single-passage steady-state model simplifies the complex circumferential interactions inherent in tandem impellers, the acceptor-generator mechanism of the tandem gap quantified by the mass flow averaged metrics provides a robust indicator that the optimized performance trends are physically grounded.

3.2. Internal Flow Field Analysis

To provide a deeper understanding of the aerodynamic performance of the optimized design, the internal flow field was investigated using pressure contours, velocity vectors, and relative Mach number distributions. This analysis helps to identify the physical reasons behind the performance gains and characterizes the behavior of the working fluid within the blade channels.

The static pressure distribution across the flow domain, as illustrated in Figure 7 shows a progressive and uniform pressure rise from the inlet (approximately 100 kPa) toward the outlet (approximately 128 kPa). The smooth transition of the pressure contours suggests a stable compression process and efficient energy transfer from the blades to the fluid. No significant pressure fluctuations or adverse pressure gradients were detected in the primary flow path, indicating that the optimized blade geometry maintains a healthy pressure loading.

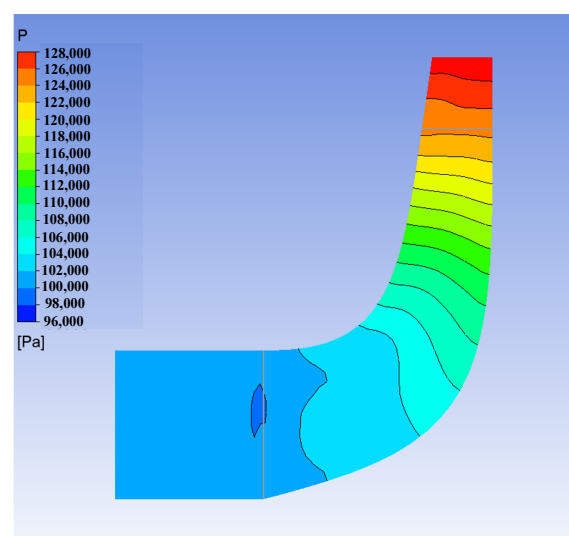


Figure 7. Static pressure distribution across the computational domain.

The velocity field and streamlines in the blade-to-blade view, presented in Figure 8, reveal that the flow is generally well-aligned with the blade curvature. This alignment demonstrates that the optimized blade angles are well-suited for the design mass flow rate. However, a localized reduction in velocity, indicated by the blue regions, is observed on the suction side near the trailing edge. These regions suggest a slight thickening of the boundary layer or incipient flow separation, which is a common source of aerodynamic loss in turbomachinery. To complement this qualitative observation with quantitative evidence, the mass-flow-averaged total pressure was evaluated at the impeller inlet and outlet for both the baseline and optimised configurations. For the baseline impeller, the total pressure decreases from 101,315 Pa at the inlet to 100,075 Pa at the outlet, corresponding to a total pressure loss of approximately 1.22%. In contrast, the optimised tandem configuration exhibits a pressure reduction from 101,314 Pa to 99,095.4 Pa, resulting in a total pressure loss of approximately 2.19%. Although the tandem configuration shows a higher local rotor loss due to stronger flow turning and enhanced diffusion, it produces a more uniform exit flow with reduced separation and weakened wake structures. This behaviour leads to improved downstream pressure recovery and contributes to the higher overall isentropic efficiency observed for the optimised design, thereby providing quantitative support for the boundary layer re-energisation effect of the tandem blade configuration. Despite the increased local rotor losses associated with stronger flow turning, the core flow remains energetic and well-directed toward the exit. It should be noted that the interpretation of the internal

flow features discussed above is based on a qualitative assessment of the velocity, pressure, and Mach number fields. While the observed flow patterns are consistent with boundary layer re-energisation and reduced separation, no explicit entropy generation analysis, secondary flow kinetic energy evaluation, or vortex-identification criteria (e.g., Q-criterion) have been performed in the present study. The primary purpose of the flow visualisation is to support the observed aerodynamic performance trends rather than to provide a detailed quantitative decomposition of individual loss mechanisms. A more comprehensive loss analysis using entropy-based or vorticity-based metrics is therefore identified as a valuable direction for future work [23].

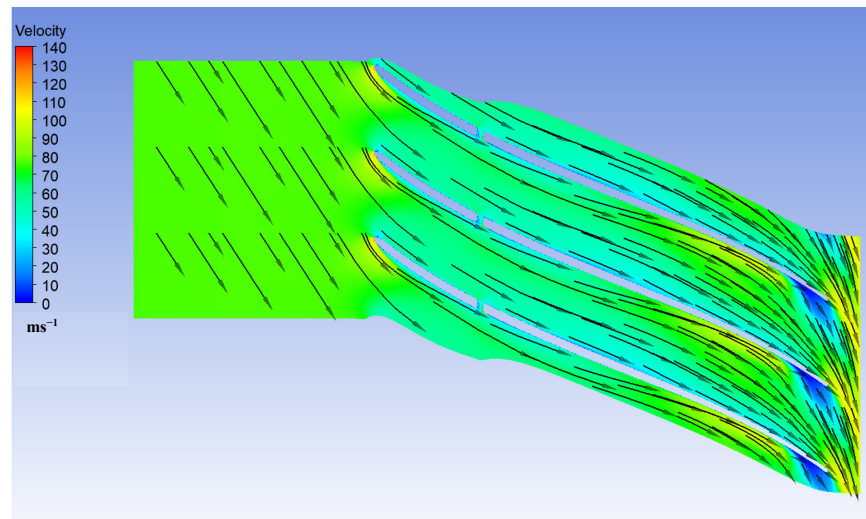


Figure 8. Velocity vectors and streamlines showing the flow behavior between the blades.

Furthermore, Figure 9 displays the distribution of the relative Mach number (M_{rel}) within the blade channels. The flow remains entirely subsonic throughout the domain, with a peak relative Mach number of approximately 0.38. Clearly defined stagnation points, where the relative Mach number approaches zero, are observable at the leading edges of the blades. This accurately captures the stagnation physics and affirms the aerodynamic efficiency of the current design. The gradual acceleration of the fluid over the blade surfaces, coupled with the absence of shock waves or abrupt velocity changes, further substantiates the effectiveness of the optimized geometry.

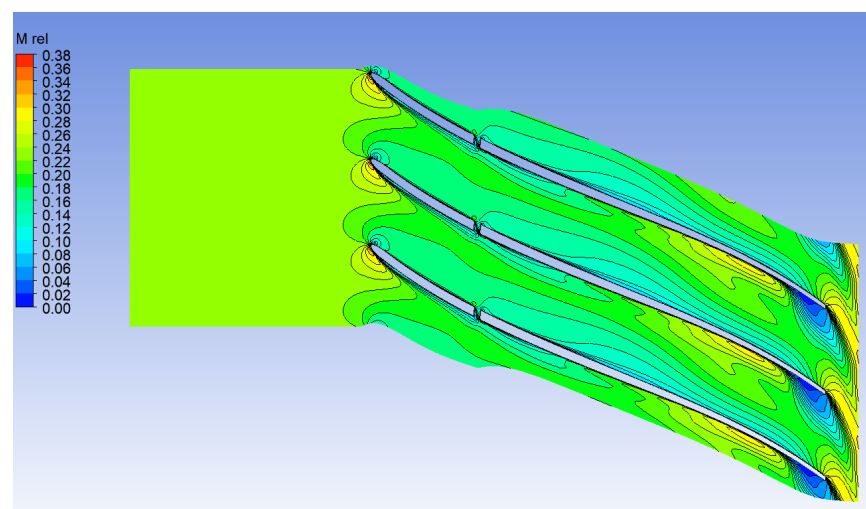


Figure 9. Relative Mach number distribution at the mid-span section.

3.3. Sensitivity Analysis Results

Figure 10 shows the local sensitivity analysis for the tandem blade compressor's performance. This analysis assesses how performance objectives respond to changes near a specific reference design point on the response surface. The findings indicate that aerodynamic performance depends on both operating conditions and critical geometric parameters. The reported sensitivity percentages are obtained by normalizing the absolute values of the local sensitivity coefficients with respect to the maximum sensitivity magnitude among all parameters, enabling a consistent relative comparison of the influence of each design variable. The sensitivity analysis is evaluated around the baseline impeller geometry, which is defined as the reference design point for the response surface [24].

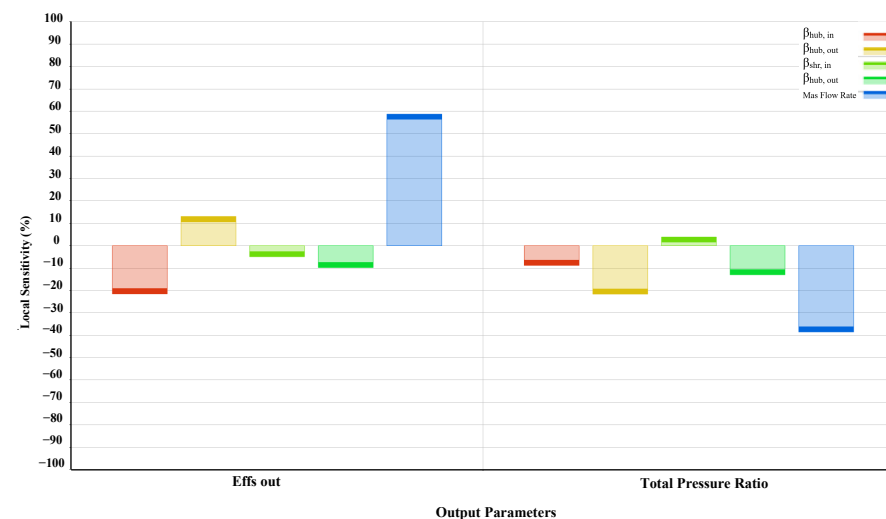


Figure 10. Local Sensitivity Analysis of Efficiency and Total Pressure Ratio.

3.3.1. Sensitivity of Isentropic Efficiency

Concerning the isentropic efficiency, the mass flow rate is identified as the primary factor, demonstrating a notable positive sensitivity of approximately 60%. This suggests that, at the prevailing operating point, an increase in the mass flow rate causes a shift in the compressor toward a higher efficiency domain.

Among the geometric parameters, the hub inlet angle exhibits a considerable negative influence ($\sim -22\%$), indicating that an increase in the blade angle at the hub leading edge results in efficiency penalties, likely due to heightened incidence losses. Conversely, the hub outlet angle shows a positive sensitivity ($\sim 13\%$), underscoring its potential role in optimizing flow behavior at the blade exit to enhance overall efficiency.

3.3.2. Sensitivity of Total Pressure Ratio

The sensitivity results for the total pressure ratio align with established compressor characteristic curves. The mass flow rate exhibits a significant negative sensitivity ($\sim -38\%$), thereby corroborating the negative slope of the pressure-rise characteristic. The hub outlet angle is identified as the most critical geometric parameter influencing pressure generation, with an approximate negative sensitivity of 22%. This correlation suggests that increasing the hub outlet angle (i.e., increasing the backsweep) results in a reduction in the total pressure ratio. Such a relationship represents a well-documented design trade-off, where pressure ratio may be sacrificed to potentially improve the stable operating range and stall margin. Furthermore, the shroud outlet angle demonstrates a notable negative effect ($\sim -13\%$), whereas the shroud inlet angle is the sole geometric parameter exerting a slight positive influence on pressure generation. Although the mass flow rate exhibits a measurable influence on the performance metrics, the interpretation of the sensitivity

results is deliberately focused on the geometric parameters. The mass flow rate is therefore treated as an operating parameter that provides a physically realistic flow context, rather than as a parameter guiding the geometric optimisation process [25].

3.4. Combined Effect of Mass Flow and Blade Angles

The interaction between the operational parameter mass flow rate and the most critical geometric parameter hub outlet angle on the total pressure ratio is illustrated in the 3D response surface plot in Figure 11.

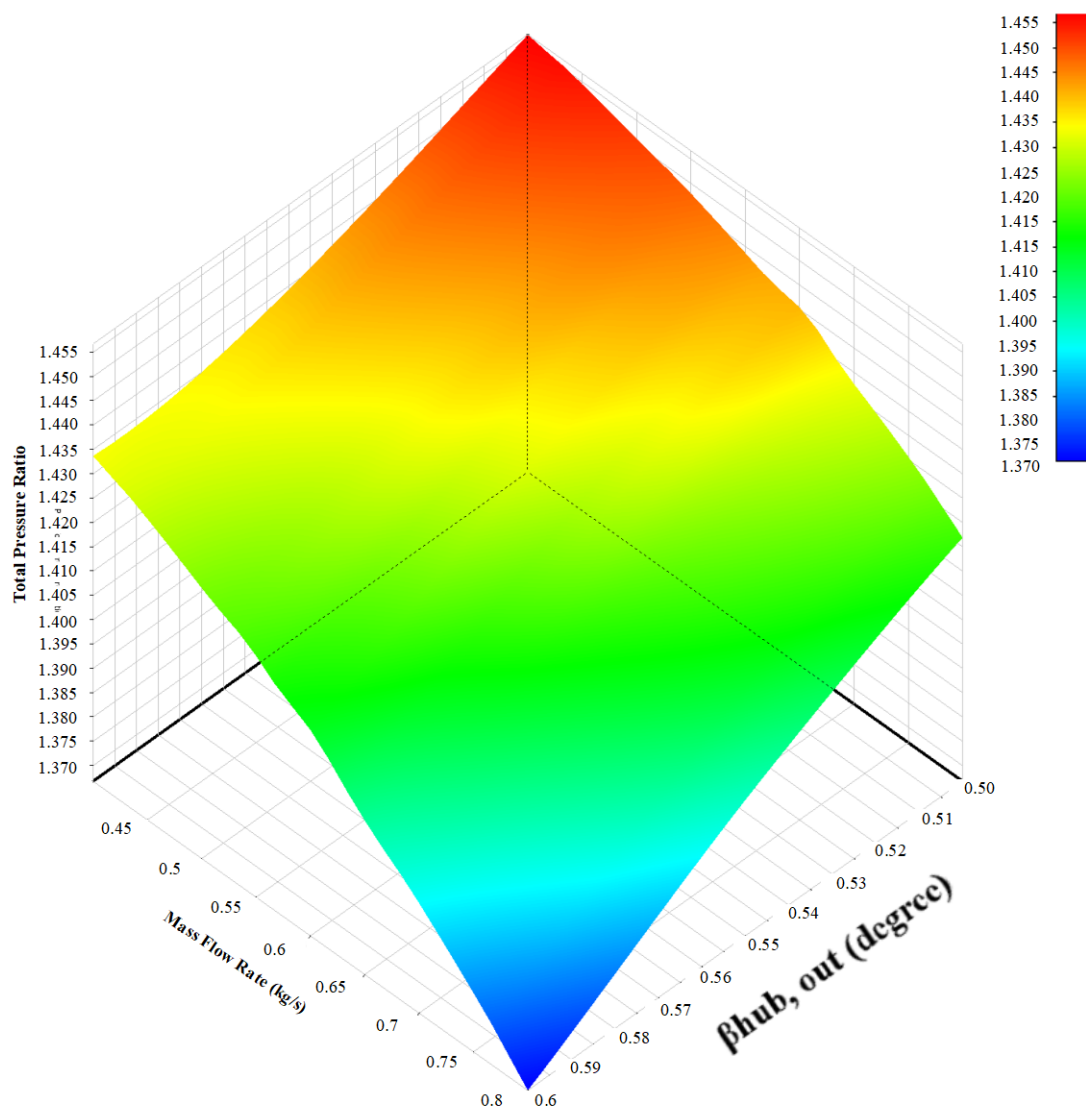


Figure 11. Three-dimensional response surface plot showing the combined effect of Mass Flow Rate and Hub Outlet Angle on Total Pressure Ratio.

The surface demonstrates a distinct monotonic gradient, thereby confirming the predominant influence of the mass flow rate on the pressure rise capability. As observed, the region of maximum total pressure ratio (indicated in red, PR approximately 1.45) is situated at the lower bound of the mass flow rate ($m = 0.4 \text{ kg/s}$) and the lower bound of the hub outlet angle ($\beta_{hub, out} = 50^\circ$). Progressing along the mass flow axis towards the design point ($m = 0.8 \text{ kg/s}$), a consistent decline in the pressure ratio is evident. Concurrently, an increase in the hub outlet angle from 50° to 60° results in a secondary reduction in the pressure ratio. This behavior indicates that, for this particular tandem impeller, lower

hub outlet angles are advantageous for maximizing the pressure head, although this may potentially compromise the width of the operating range.

3.5. Multi-Objective Optimization Results

The aerodynamic optimization of the tandem blade centrifugal compressor was performed using a MOGA within the ANSYS Workbench environment. The optimization problem was formulated to simultaneously maximize the total-to-total Pressure Ratio (PR) and the isentropic efficiency (η), which are inherently competing performance objectives in centrifugal compressor design.

To preserve the natural trade-off between these objectives and to avoid subjective bias, no explicit weighting factors were assigned. Instead, a Pareto-based optimization strategy was employed, allowing the MOGA to generate a set of non-dominated solutions representing different compromises between pressure rise and aerodynamic efficiency. The multi-objective optimization process was initiated based on the design space and constraints defined in Section 2.5.

The Pareto frontier obtained from the MOGA optimization is illustrated in Figure 12, where the isentropic efficiency is plotted against the total-to-total pressure ratio for all feasible design candidates.

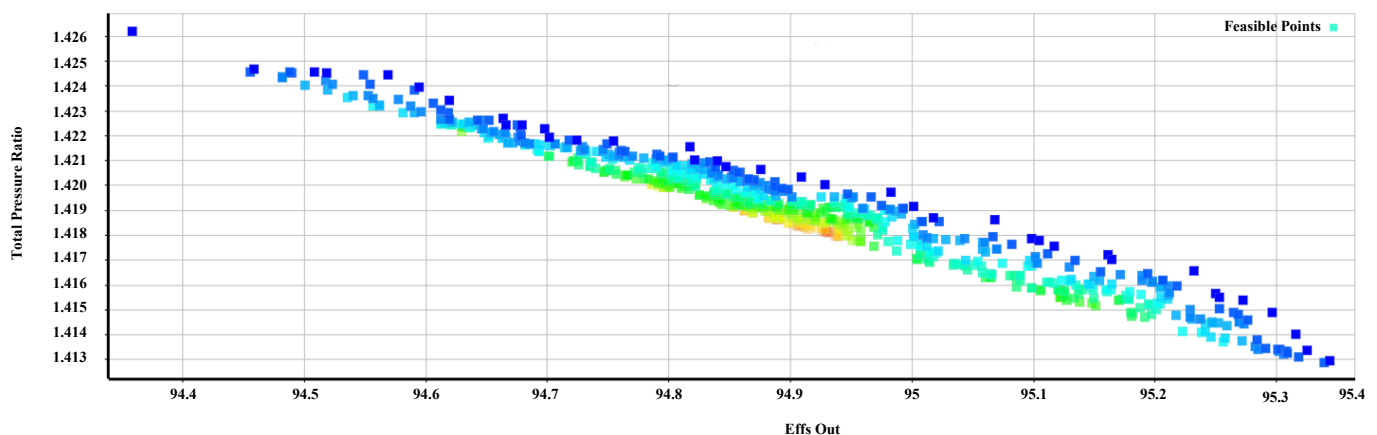


Figure 12. Pareto frontier obtained from the MOGA optimization illustrating the trade-off between isentropic efficiency and total-to-total pressure ratio.

The distribution of solutions reveals a clear inverse relationship between the two objectives, confirming the trade-off nature of the optimization problem. Designs located at the extremes of the Pareto frontier favor either pressure ratio or efficiency at the expense of the other, while intermediate solutions provide a balanced aerodynamic performance.

The final optimal design was selected from the Pareto frontier as a compromise solution that achieves a balanced improvement in both objectives rather than maximizing a single performance metric. This selection avoids extreme solutions that could lead to excessive efficiency loss or insufficient pressure rise, and ensures robust aerodynamic performance under practical operating conditions.

The aerodynamic performance of the selected optimal tandem impeller is quantitatively compared with a reference impeller reported in the literature in Table 3.

As summarized in Table 3, the optimized tandem impeller configuration exhibits a clear improvement in total-to-total pressure ratio compared to the baseline design while operating at nearly the same mass flow rate. The pressure ratio increases from 1.306 to 1.416, corresponding to an improvement of approximately 8.47%, which demonstrates the effectiveness of the applied optimization strategy in enhancing the pressure-rise capability of the impeller.

Table 3. Comparison of Baseline and Optimized Design Parameters.

Parameter	Baseline Design	Optimized Design	Change (%)
Blade Count (Z)	18	22	22.22
$\beta_{\text{hub,in}}$ (degree)	0	−4.98	-
$\beta_{\text{hub,out}}$ (degree)	55	53.59	−2.56
$\beta_{\text{shr,in}}$ (degree)	59.33	61.04	+2.88
$\beta_{\text{shr,out}}$ (degree)	44.99	40.79	−9.33
Mass Flow Rate (kg/s)	0.789	0.79	+0.13
Total Pressure Ratio	1.306	1.416	+8.47

The geometric modifications associated with this improvement primarily involve an increase in blade count and targeted adjustments of the blade angles at both the hub and shroud. For angular parameters, percentage changes are reported relative to the baseline values given in Table 3. No percentage change is reported for the hub inlet angle since the baseline value is zero, and the original sign convention of blade angles is preserved for all configurations. In particular, the increase in blade count from 18 to 22 contributes to reduced slip effects and improved flow guidance, while the moderate reductions in the hub and shroud outlet angles support more controlled diffusion within the blade passages. The small variation in mass flow rate confirms that the performance gain is achieved without a significant shift in operating condition.

Overall, the results reported in Table 3 indicate that the optimal design identified from the Pareto frontier represents a balanced aerodynamic improvement in terms of pressure ratio, achieved through physically consistent geometric modifications rather than extreme changes in operating parameters.

The optimal configuration corresponds to a tandem impeller with 22 blades, achieving a total-to-total pressure ratio of 1.416 and a peak isentropic efficiency of 95.23%.

Although the optimal blade count identified by the MOGA corresponds to the upper bound of the investigated range, this outcome does not indicate an unreasonable or artificially constrained design space. The upper limit of the blade count was deliberately selected based on aerodynamic, mechanical, and manufacturing considerations reported in the literature for tandem centrifugal impellers. Increasing the blade count beyond this range is known to significantly increase flow blockage, frictional losses, and structural complexity, which can offset potential gains in pressure rise and efficiency. Therefore, the convergence of the optimal solution toward this boundary reflects a physically meaningful aerodynamic trend within the admissible design space rather than a limitation of the optimisation setup.

The findings obtained, as in other studies in the literature, show that increasing the number of blades and using a properly designed tandem propeller geometry improves flow control, reduces slip effects, and increases the pressure ratio [24,25]. Furthermore, CFD-based optimization studies have shown that convergence towards a higher number of blades can provide net performance gains despite increased friction due to reduced separation and diffusion losses [26]. Furthermore, the slight decrease in the negative hub inlet angle and hub outlet angle is consistent with the loss mechanisms described by Denton [7] and Bardelli et al. [26], as such changes reduce event-related and secondary flow losses in the hub region. In summary, the increase in total pressure ratio observed at nearly constant mass flow, combined with high peak isentropic efficiency, is in good agreement with the established aerodynamic performance improvement trends reported for tandem-bladed centrifugal compressor systems.

4. Conclusions

This study investigates and optimises the aerodynamic performance of a tandem blade centrifugal compressor utilising a parametric CFD approach coupled with RSM. The integration of geometric variables with operational conditions furnishes a comprehensive understanding of the compressor's behavior. The principal findings of this research are summarized as follows:

1. Local sensitivity analysis identified the hub outlet angle as the primary geometric parameter influencing the total pressure ratio, exhibiting a strong negative correlation attributable to the backsweep effect. Conversely, the hub inlet angle was determined as the critical factor affecting isentropic efficiency by directly impacting incidence losses at the blade root.
2. A detailed flow field analysis, through velocity, pressure, and relative Mach number contours, validated the aerodynamic advantages of the tandem configuration. The inducer-exducer interface effectively re-energises the boundary layer, thereby suppressing flow separation and ensuring a more uniform velocity distribution relative to conventional designs.
3. The operational parameter of mass flow rate demonstrated a predominant influence on both performance metrics. This underscores the importance of incorporating operational conditions within the geometric optimization process to accurately delineate the peak-efficiency regions of the compressor map.
4. The MOGA effectively explored the design space, revealing that a blade count of 22 optimises flow guidance, notably reducing secondary flow losses without imposing excessive penalties due to skin friction.
5. The final optimized design exhibited outstanding performance, attaining an isentropic efficiency of 95.23% and a total pressure ratio of 1.416 at a mass flow rate of 0.79 kg/s, representing a significant enhancement over the baseline configuration.

These findings substantiate that the tandem blade concept, when synergized with optimized blade angles and count, constitutes a superior solution for high-efficiency compression systems, offering enhanced pressure-rise capabilities and improved flow stability.

Author Contributions: Conceptualization, S.Ö.; methodology, S.Ö. and M.E.S.; software, M.E.S.; investigation, S.Ö. and M.E.S.; writing—original draft preparation, S.Ö. and M.E.S.; writing—review and editing, S.Ö. and M.E.S.; visualization, M.E.S.; supervision, S.Ö. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The original contributions presented in this study are included in the article material. Further inquiries can be directed to the corresponding authors.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

CFD	Computational Fluid Dynamics
RSM	Response Surface Methodology
RANS	Reynolds-Averaged Navier–Stokes
MOGA	Multi-Objective Genetic Algorithm
DOE	design of experiments
RMS	root-mean-square
RMSE	Root Mean Square Error
PR	Pressure Ratio

η	isentropic efficiency
ρ	fluid density
U	velocity vector
p	pressure
S_M	The viscous stress tensor
λ	thermal conductivity
S_E	denotes energy sources
SST	The Shear Stress Transport
k	turbulent kinetic energy
ω	specific dissipation rate
β	blade metal angle
Z	primary blade count

References

1. Shaaban, S. Design optimization of a centrifugal compressor vaneless diffuser. *Int. J. Refrig.* **2015**, *60*, 142–154. [\[CrossRef\]](#)
2. Demir, F.; Özer, S.; Demir, U.; Körükçü, K.; Oduncu, H.; Ekin, M.Ş. Design optimization and field validation of industrial fans with CFD for cement production: Performance, energy savings, and environmental benefits. *Sustainability* **2025**, *17*, 10279. [\[CrossRef\]](#)
3. Hagelstein, D.; Hillewaert, K.; van den Braembussche, R.A.; Engeda, A.; Keiper, R.; Rautenberg, M. Experimental and Numerical Investigation of the Flow in a Centrifugal Compressor Volute. *J. Turbomach.* **1999**, *122*, 22–31. [\[CrossRef\]](#)
4. Galloway, L.; Rusch, D.; Spence, S.; Vogel, K.; Hunziker, R.; Kim, S.I. An investigation of centrifugal compressor stability enhancement using a novel vaned diffuser recirculation technique. *J. Turbomach.* **2018**, *140*, 121009. [\[CrossRef\]](#)
5. Cheng, H.; Song, Z.; Liu, B. Aerodynamic Study on the Influence Mechanism of Bow Blades on the Flow Field of Supersonic Tandem Rotors. *Energies* **2022**, *15*, 4474. [\[CrossRef\]](#)
6. Josuhn-Kadner, B. Flow field and performance of a centrifugal compressor rotor with tandem blades of adjustable geometry. In *Turbo Expo: Power for Land, Sea and Air*; American Society of Mechanical Engineers: New York, NY, USA, 1994; Volume 78835, p. V001T01A004.
7. Denton, J.D. Loss mechanisms in turbomachines. *J. Turbomach.* **1993**, *115*, 621–656. [\[CrossRef\]](#)
8. Jin, W.; Wang, J.; Li, J.; Xu, R.; Zhou, M.; Huang, Q. Optimization Design of Multi-Blade Centrifugal Fan Based on Variable Weight PSO-BP Prediction Model and Multi-Objective Beluga Optimization Algorithm. *Appl. Sci.* **2025**, *15*, 5950. [\[CrossRef\]](#)
9. Söylemez, M.E.; Behçet, R.; Parlak, Z. Analysis and Optimization of the Performances of the Tandem Blade Radial Compressor Using the CFD. *Appl. Sci.* **2024**, *14*, 4256. [\[CrossRef\]](#)
10. Ziliang, L.; Xingen, L.; Ge, H.; Enliang, H.; Chengwu, Y.; Junqiang, Z. Numerical and experimental investigation of flow mechanism and application of tandem-impeller for centrifugal compressor. *Aerosp. Sci. Technol.* **2020**, *100*, 105819. [\[CrossRef\]](#)
11. Guo, Y.; Wang, Y.; Cao, Y.; Long, Z. A New Optimization Design Method of Multi-Objective Indoor Air Supply Using the Kriging Model and NSGA-II. *Appl. Sci.* **2023**, *13*, 10465. [\[CrossRef\]](#)
12. Roberts, D.A.; Kacker, S.C. Numerical Investigation of Tandem-Impeller Designs for a Gas Turbine Compressor. *J. Turbomach.* **2002**, *124*, 36–44. [\[CrossRef\]](#)
13. Khiloshiya, M.; Mouvanal, S.; Jayakumar, R. CFD Based Design of a Centrifugal Compressor for Enhanced Range of Operation. In Proceedings of the ASME Turbo Expo 2025: Turbomachinery Technical Conference and Exposition, Memphis, TN, USA, 16–20 June 2025; p. V012T34A009. [\[CrossRef\]](#)
14. Guidotti, E. Towards Centrifugal Compressor Stages Virtual Testing. Ph.D. Thesis, University of Bologna, Bologna, Italy, 2014. Available online: https://amsdottorato.unibo.it/id/eprint/6550/1/Guidotti_Emanuele_tesi.pdf (accessed on 1 February 2026).
15. Yoon, S.Y.; Lin, Z.; Lim, K.T.; Goyne, C.; Allaire, P.E. Model validation for an AMB-based compressor surge control test rig. In Proceedings of the 48th IEEE Conference on Decision and Control, Shanghai, China, 15–18 December 2009; IEEE: New York, NY, USA, 2009; pp. 756–761.
16. ANSYS, Inc. ANSYS CFX Theory Guide. ANSYS Help. 2017. Available online: <https://www.ansys.com> (accessed on 1 February 2026).
17. Mojaddam, M.; Moussavi Torshizi, S.A. Design and optimization of meridional profiles for the impeller of centrifugal compressors. *J. Mech. Sci. Technol.* **2017**, *31*, 4853–4861. [\[CrossRef\]](#)
18. Juan-Nan, L.; Dingxi, W.; Xiuquan, H. Investigation of turbulence models for aerodynamic analysis of a high-pressure-ratio centrifugal compressor. *Phys. Fluids* **2023**, *35*, 106108. [\[CrossRef\]](#)
19. ASME. ASME V&V 20–2009; Standard for Verification and Validation in Computational Fluid Dynamics and heat TRANSFER. American Society of Mechanical Engineers: New York, NY, USA, 2009. Available online: <https://www.asme.org/getmedia/692ee706-d396-49fd-b62c-d8e3de655a27/21356.pdf> (accessed on 1 February 2026).

20. Mangani, L.; Casartelli, E.; Mauri, S. Flow field investigation in a high pressure ratio centrifugal compressor with vaneless diffuser. In Proceedings of the ASME Turbo Expo 2011: Turbine Technical Conference and Exposition, Vancouver, BC, Canada, 6–10 June 2011; ASME: New York, NY, USA, 2011. [\[CrossRef\]](#)
21. Xu, C.; Amano, R. Aerodynamic and structure considerations in centrifugal compressor design blade lean effects. In Proceedings of the ASME Turbo Expo, Copenhagen, Denmark, 11–15 June 2012; p. 68207.
22. Qin, R.; Ju, Y.; Galloway, L.; Spence, S.; Zhang, C. High Dimensional Matching Optimization of Impeller–Vaned Diffuser Interaction for a Centrifugal Compressor Stage. *ASME J. Turbomach* **2020**, *142*, 121004. [\[CrossRef\]](#)
23. Shao, X.; Liu, Y.; Wang, B.; Li, X.; Chen, J.; Zhu, Z.; Ma, X. Fast regulation of multi-position differentiated environment: Multi-step joint optimization of air supply parameters. *Build. Environ.* **2023**, *239*, 110425. [\[CrossRef\]](#)
24. Ju, Y.; Qin, R.; Kipouros, T.; Parks, G.; Zhang, C. A high-dimensional design optimisation method for centrifugal impellers. *Proc. Inst. Mech. Eng. Part A J. Power Energy* **2016**, *230*, 272–288. [\[CrossRef\]](#)
25. Bernhard, S.; Mihai, M. Flow phenomena leading to surge in a centrifugal compressor. *Energy* **2016**, *103*, 572–587. [\[CrossRef\]](#)
26. Bardelli, M.; Cravero, C.; Marini, M.; Marsano, D.; Milingi, O. Numerical Investigation of Impeller–Vaned Diffuser Interaction in a Centrifugal Compressor. *Appl. Sci.* **2019**, *9*, 1619. [\[CrossRef\]](#)

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.