Çukurova Üniversitesi
Mühendislik Fakültesi Dergisi
Çukurova University
Journal of the Faculty of Engineering
ISSN: 2757-9255 CİLT/VOLUME: 41 SAYI/ISSUE: 3 EYLÜL/SEPTEMBER 2026

Optimization of Centrifugal Fan Efficiency Based on Volute Meridional Inlet Width and Stepanoff Number Using CFD and Kriging-Based Response Surface Method

Melih Gazi KIRPIK [1,a], **Hüseyin AKILLI** [1,b], **Tuğrul SAPMAZ** [1,c], **Hamit TÜRKMEN** [2,d]

[1]*Çukurova University, Engineering Faculty, Mechanical Engineering Department, Adana, Türkiye*
[2]*Çukurova Silo Company, Adana, Türkiye*

[a]***ORCID***: *0000-0002-8011-6331;* [b]***ORCID***: *0000-0002-5342-7046;* [c]***ORCID***: *0000-0002-3552-1952;*
[d]***ORCID***: *0000-0002-3084-0890*

Article Info
Received : 18.11.2025
Accepted : 22.09.2026
DOI: 10.21605/cukurovaumfd.1826186

Corresponding Author
Melih Gazi KIRPIK
melihgazi_kirpik@hotmail.com

Keywords
Centrifugal fan
Fan characteristic curve
Computational fluid dynamics
Ansys CFX
Response surface methodology

How to cite: *KIRPIK, M.G., AKILLI, H., SAPMAZ, T., TÜRKMEN, H., (2026). Optimization of Centrifugal Fan Efficiency Based on Volute Meridional Inlet Width and Stepanoff Number Using CFD and Kriging-Based Response Surface Method. Cukurova University, Journal of the Faculty of Engineering, 41(3), 737-746.*

ABSTRACT

The present work optimizes the efficiency of a centrifugal fan by applying Computational Fluid Dynamics and a Kriging-based response surface methodology. The aim is to improve fan efficiency in the low to medium pressure values while maintaining the existing broad pressure range performance. Volute meridional inlet width and Stepanoff number were chosen as variables. Their effects on total efficiency and volume flow rate were evaluated by applying incompressible steady state Ansys CFX simulations after mesh independence confirmation. A Design of Experiments framework at nine design points was generated by using Latin Hypercube Sampling method. A kriging metamodel was then generated and validated at 7 points with a relative average absolute error lower than 3%. The optimum design was found at a volute meridional inlet width of about 0.27 m and a Stepanoff number of 0.3. The optimized geometry results in an average relative efficiency improvement of approximately 6.79% in the 0–900 Pa range.

CFD ve Kriging Tabanlı Tepki Yüzeyi Yöntemi Kullanılarak Volüt Meridyonel Giriş Genişliği ve Stepanoff Sayısına Dayalı Santrifüj Fan Verimliliğinin Optimizasyonu

Makale Bilgileri
Geliş : 18.11.2025
Kabul : 22.09.2026
DOI: 10.21605/cukurovaumfd.1826186

Sorumlu Yazar
Melih Gazi KIRPIK
melihgazi_kirpik@hotmail.com

Anahtar Kelimeler
Santrifüj fan
Fan karakteristik eğrisi
Hesaplamalı akışkanlar dinamiği
Ansys CFX
Tepki yüzeyi metodolojisi

Atıf şekli: *KIRPIK, M.G., AKILLI, H., SAPMAZ, T., TÜRKMEN, H., (2026). CFD ve Kriging Tabanlı Tepki Yüzeyi Yöntemi Kullanılarak Volüt Meridyonel Giriş Genişliği ve Stepanoff Sayısına Dayalı Santrifüj Fan Verimliliğinin Optimizasyonu. Çukurova Üniversitesi, Mühendislik Fakültesi Dergisi, 41(3), 737-746.*

ÖZ

Bu çalışma, Hesaplamalı Akışkanlar Dinamiği ve Kriging tabanlı tepki yüzeyi yöntemi kullanılarak bir santrifüj fanın verimliliğini optimize etmektedir. Amaç, mevcut geniş basınç aralığı performansını korurken, düşük ila orta basınç aralığında fan verimliliğini artırmaktır. Tasarım değişkenleri olarak salyangoz gövde meridyonel giriş genişliği ve Stepanoff sayısı seçilmiştir. Bu değişkenlerin toplam verimlilik ve hacimsel debi üzerindeki etkileri, ağ bağımsızlığı doğrulamasının ardından sıkıştırılamaz akış ve kararlı rejim koşullarında gerçekleştirilen Ansys CFX simülasyonları ile değerlendirilmiştir. Latin Hiperküp Örnekleme yöntemi kullanılarak dokuz tasarım noktasını kapsayan bir Deney Tasarımı oluşturulmuştur. Daha sonra bir Kriging meta modeli oluşturulmuş ve %3'ten düşük bir bağıl ortalama mutlak hata değeri ile 7 noktada doğrulanmıştır. Optimum tasarım; yaklaşık 0,27 m salyangoz gövde meridyonel giriş genişliği ve 0,3 Stepanoff sayısı değerlerinde elde edilmiştir. Optimize edilen geometri, 0–900 Pa aralığında ortalama bağıl verimlilikte yaklaşık %6,79'luk bir iyileşme sağlamaktadır.

1. INTRODUCTION

Turbomachinery is a kind of mechanical device which is designed to exchange energy between a moving fluid and a rotating component. Centrifugal fan is one of the most common turbomachinery types used in industry. A centrifugal fan operates on the principle of imparting energy to air or gas through the centrifugal force produced by a rotating impeller. Air enters the fan housing axially at the inlet and is then accelerated radially outward along the blades. The resulting increase in velocity is converted into pressure energy as the airflow passes through the scroll casing. This process transforms mechanical energy from the motor that impeller is connected into pressure and kinetic energy within the fluid stream [1]. A centrifugal fan example is given in Figure 1.

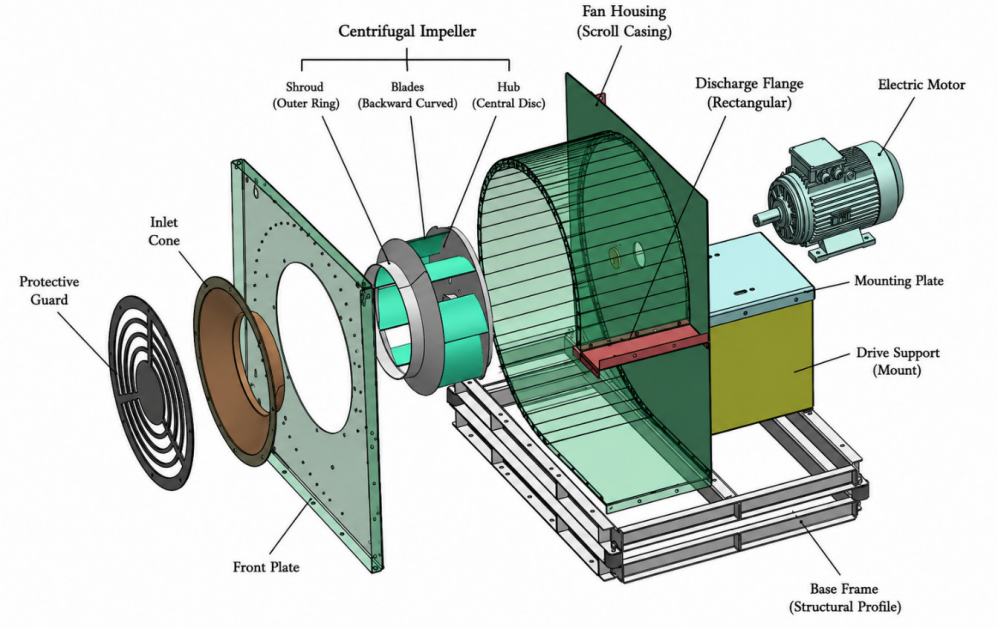


Figure 1. Centrifugal fan

Electric motor systems are one of the main electricity consumption devices in industry, with fans constituting one of the major energy consuming groups among these. Although fan systems are theoretically capable of achieving a total efficiency exceeding 79%, it has been reported that average operating efficiencies in the field can be less than 50% and under certain conditions may fall to around 15–20%. Hence, the improvement of a fan system has great potential to improve energy efficiency with substantial benefit for industrial use [2]. Optimization of different fan systems can be achieved by using CFD tools. Computational Fluid Dynamics (CFD) is a vital tool in the design and optimization of centrifugal fans, allowing engineers to model complex interactions between airflow and structural components with high precision. By simulating pressure distributions, velocity fields, and stress concentrations for the fan system, CFD enables the identification of performance bottlenecks and potential sources of energy loss. This predictive capability facilitates informed design modifications, such as blade geometry design, casing adjustments and sizing, or hub optimization, ultimately improving aerodynamic efficiency and operational reliability. Consequently, integrating CFD into the design process not only improves energy efficiency but also reduces the risk of mechanical failure, supporting economic and environmental objectives in industrial applications. Meng et al, conducted an optimization study to investigate the three-dimensional viscous flow characteristics within a centrifugal fan which is equipped with backward-curved blades. The research focuses on how variations in the inlet angle, outlet blade angle, and the number of blades affect the aerodynamic performance of the assembly of impeller and volute together [3]. In order to improve the lift-to-drag ratio, Wu and Huang's study led to the development of an optimized airfoil series. This newly created profile was then used in the construction of fan blades. Comprehensive three-dimensional numerical simulations were carried out in Ansys CFX using an axial fan as the research model [4]. The aerodynamic performance of axial fans equipped with three different blade configurations was analyzed and compared with each other. In the study of Jung et al, it was investigated the optimization of diffuser design in small, high-speed centrifugal fans by using CFD and Design of Experiments (DOE) methods [5]. As a result, it was found that the trailing edge (TE β) angle of the guide

vane has the greatest impact on fan performance. From this study it was concluded that selecting the optimal angle and blade geometry significantly improves the fan's vacuum (suction) capacity and efficiency. In this study, it was also shown that the CFD result was consistent with experimental data, confirming that the proposed diffuser design enhances performance. Some other studies which were conducted by other scientists [6-9] can be given as examples which CFD method was used for centrifugal fan or pump applications.

For improving fan efficiency CFD method is a robust and effective approach that is used for this purpose commonly. Components of a centrifugal fan such as impeller, volute or extension can be considered individually to improve efficiency by considering each parameter that affects efficiency. However, since changes in one parameter affect the overall flow behavior, focusing on a single parameter may lead to suboptimal design decisions. Therefore, the interaction between design parameters must be considered, and they should be optimized simultaneously to achieve better efficiency optimization results. In order to maximize performance improvement, centrifugal fans should be viewed as a single, integrated system in which the flow field is assessed holistically. The purpose of this study is to increase the centrifugal fan's efficiency by using CFD and response surface methods together. Even small increases in fan efficiency can result in significant energy cost savings because the fan runs most of the time to maintain the necessary environmental conditions for grain storage in silo bins under certain climate conditions. A fan model was created by using a fan design program and in order to determine the characteristic curve of the fan and optimization, CFX simulations were generated. Stepanoff number together with meridional inlet width of volute were considered as design variables for the volute geometry to increase the efficiency of the whole system. Design of Experiments (DOE) was conducted to analyze the effect of design variables at different consideration points. Response Surface Method (RSM) is a simulation-based design technique that enables researchers to construct predictive models, assess the influence of multiple factors, identify optimal conditions for desired responses, and simultaneously reduce the number of required simulations [3]. The main contribution of this study lies in using response surface method with kriging metamodel to optimize the centrifugal fan volute to increase total efficiency and find out the optimum values for design variables. In conclusion, characteristic curves of the fans before and after optimization were compared to each other and optimized system outperformed the prototype system under desired working conditions.

2. MATERIAL AND METHODS

2.1. Centrifugal Fan Characteristic Properties

Prototype centrifugal fan model consists of 3 components. These are stator, impeller and volute with its extension. Characteristic properties of these components are given in Table 1.

Table 1. Characteristic properties of centrifugal fan components

Components	Characteristic properties	Values before optimization	Values after optimization
Impeller	Impeller diameter	632 mm	632 mm
	Leading edge width	154.5 mm	154.5 mm
	Outlet width	117 mm	117 mm
	Suction diameter	335 mm	335 mm
	Number of blades	7	7
	Blade types	Circular	Circular
	Circular blade radius	281.9 mm	281.9 mm
	Rotational speed	1500 RPM	1500 RPM
Stator	Length	450 mm	450 mm
	Diameter	335 mm	335 mm
Volute	Stepanoff number	0.4	0.3 (25% decreased)
	Meridional inlet width	250 mm	270 mm (8% increased)
	Centerline diameter	732 mm	732 mm

2.2. Computational Fluid Dynamic (CFD)

Computational Fluid Dynamics (CFD) is a numerical technique that solves the discretized governing equations of fluid flow to simulate the behavior of fluids within a defined flow domain. It enables prediction

of complex aerodynamic phenomena, evaluation of performance under different operating conditions, and optimization of design parameters with significantly reduced number of experimental requirements [10]. In this study, Ansys CFX tool was used to perform numerical simulations due to its high computational accuracy and proven validation in different applications of turbomachinery such as efficiency and volume flowrate prediction. The fan geometry was modeled by using a specialized turbomachinery design software, and Ansys CFX was employed to evaluate the mesh independence beside deriving fan characteristic curve. Through analysis of flow distribution and performance parameters, the CFD environment supported the design optimization process efficiently while maintaining reliable physical accuracy.

2.2.1. Determination of Number of Mesh Elements for Analysis

The purpose of performing a mesh independence study is to ensure that the numerical results obtained from a CFD simulation are not affected by the mesh resolution significantly. In other words, it is a way to verify that further refinement of the computational grid does not produce noticeable changes in the predicted flow parameters, such as pressure, velocity or temperature. Establishing mesh independence is essential for demonstrating the accuracy, stability, and reliability of the numerical model while avoiding unnecessary computational cost associated with excessively fine grids [11]. Centrifugal fan prototype is seen in Figure 2.

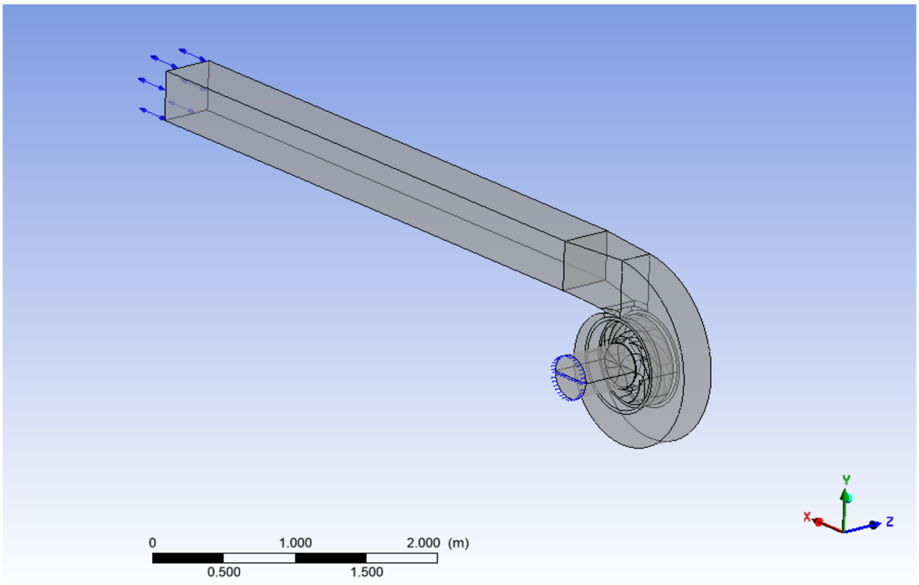


Figure 2. Prototype design of centrifugal fan

A mesh independence study was conducted to be sure that the numerical results were not affected by the grid resolution and to determine the optimal balance in between computational cost and solution accuracy. Different mesh configurations and the numbers of meshes used for the analysis are seen in Table 2.

Table 2. Number of mesh elements of centrifugal fan components for different mesh configurations

Number of mesh elements of fan components used in the mesh independence study				
	Stator	**Impeller**	**Volute and extension**	**Number of mesh elements**
Mesh 1	28005	767215	302202	1097422
Mesh 2	56624	1070220	368159	1495003
Mesh 3	145487	1107368	1016905	2269760
Mesh 4	145487	1146619	1016715	2308821

The resulting mass flow rates were 3.333 kg/s, 3.340 kg/s, 3.342 kg/s, and 3.343 kg/s, respectively, for four distinct mesh densities that ranged from roughly 1.1 million to 2.3 million elements. The relative differences in mass flow rate between successive mesh configurations were 0.21%, 0.06%, and 0.03%. The results indicate that the changes in the monitored parameter become progressively smaller as the mesh is refined and it demonstrates a clear convergence trend. Since the variation in mass flow rate became negligible with

further mesh refinement (0.03% between Mesh 3 and Mesh 4), Mesh 3 was considered sufficiently grid-independent for the present study. Table 3. shows the results of conducted study to determine the number of mesh elements to be used.

Table 3. Mass flowrates for different grid densities

Grid density vs. mass flow rate			
Test	**Number of mesh elements**	**Mass flowrate (kg/s)**	**Difference (%)**
1	1097422	3.333	
2	1495003	3.340	0.21
3	2269760	3.342	0.06
4	2308821	3.343	0.03

2.2.2. CFD Setup

Stator, impeller, and volute with its extension are three primary parts of the fan geometry taken into consideration in this study. Simulations were conducted under steady state conditions. Flow was accepted as incompressible as the maximum global velocity in stationary frame for the fluid domain is 53 m/s which corresponds to a Mach number below 0.3 under the considered conditions [12]. Figure 3. represents the velocity contour of fluid domain on a plane which is placed in the middle of the impeller in Z direction.

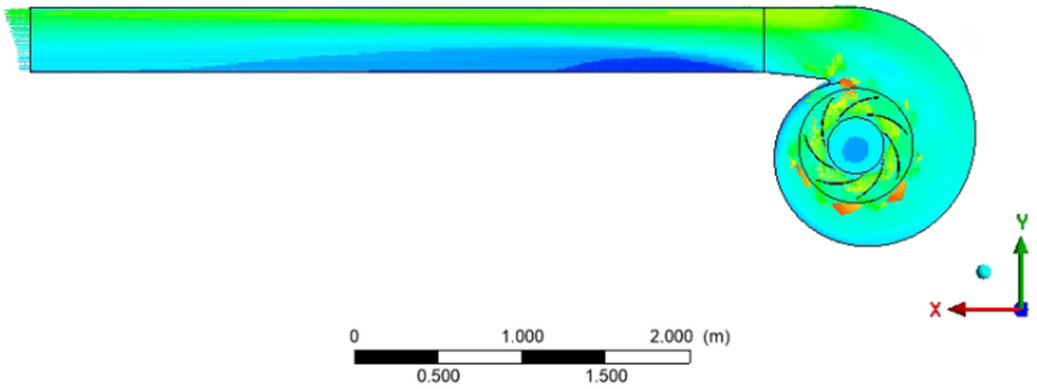


Figure 3. Stationary velocity contour of fluid domain of centrifugal fan

Both inlet and outlet are considered as opening boundary conditions as this boundary condition is appropriate for situations with possible bidirectional flow since the fluid may fully enter, fully exit, or partially do both, depending on the flow. The value of opening pressure at inlet and outlet was specified as 0 Pa. All walls were considered as smooth wall assumptions with standard roughness, and five layers of inflation were applied to the blade walls. Mesh quality metrics for the computational domain that is used for the analysis are shown in Table 4.

Table 4. Average mesh quality metrics for computational domain

Components	**Average skewness**	**Average orthogonal quality**	**Average aspect ratio**
Stator	0.267	0.732	1.94
Impeller	0.257	0.76	2.34
Volute and extension	0.232	0.766	1.841

Table 5. Mesh quality criteria and classification ranges for CFD analysis

Components	**Excellent**	**Good**	**Fair**	**Poor**	**Unacceptable**
Skewness	0-0.25	0.25-0.50	0.50-0.75	0.75-0.9	0.9-1
Orthogonal quality	1-0.9	0.90-0.75	0.75-0.50	0.5-0.25	0.25-0
Aspect ratio	1-5	5-10	10-20	20-50	50+

When the fluid domain metrics are considered based on Table 5. it can be concluded that the average values are at the range of good and excellent.

Frozen Rotor technique, which keeps the components' relative positions while altering the reference frame at the interface was used to model the interaction between the rotating and stationary components. The Frozen Rotor technique is particularly suitable for non-axisymmetric configurations, such as impeller–volute, propeller–hull, or turbine–draft tube systems, providing a robust and computationally efficient solution for geometries with a high number of blades. That's why the Frozen Rotor interfaces were applied between the inlet cone and the impeller, as well as between the impeller and the volute [13].

The upwind advection scheme, which improves numerical stability in areas with large velocity gradients by determining the value at a cell face based on upstream flow, was used for the convective terms. The upwind scheme guarantees stable convergence and works well in complex flow domains, despite being more diffusive than higher-order techniques. Combination of the upwind scheme with high-resolution turbulence numeric was selected to balance accuracy and stability. Since the primary objective of this study is to determine the overall hydrodynamic performance of system rather than the local boundary layer structure, the k-ε model combined with the Scalable Wall Function approach was deemed a suitable method in terms of accuracy, solution stability, and computational cost. Scalable wall function prevents solver inaccuracies and crashes when mesh is highly resolved near the wall. Priority of the conducted study is hydrodynamic performance rather than local wall effects. Lastly, impeller rotational speed was set to 1500 RPM as model was designed based on this rotational speed criteria.

2.3. Determining Fan Efficiency

Fan total efficiency is calculated from the formula of

$$\text{Fan total efficiency} = \frac{((Outlet\ total\ pressure(Pa) - Inlet\ total\ pressure(Pa)) \text{x} (Volume\ Flowrate\left(\frac{m^3}{s}\right)))}{((Torque\ of\ impeller\left(\frac{N}{m}\right)) \text{x} (\frac{2\pi\ RPM}{60}))} \quad [14]$$

Values of torque, volume flowrate and pressure values were taken from the results of CFX analysis to calculate the fan efficiency value.

2.4. DOE and Response Surface Optimization

Design of Experiments (DOE) provides a structured and systematic framework to investigate the influence of multiple design variables on system performance while minimizing the number of simulations. In the current study, Latin Hypercube Sampling (LHS), a statistical method that guarantees consistent and thorough coverage of the multidimensional design space, was combined with DOE. LHS does this by breaking each input variable into equally likely intervals and picking representative values from each interval. This keeps the data from clustering and makes sampling more efficient. In this study, LHS was executed using a random seed of 30 to produce a well-distributed dataset for response surface modeling, facilitating precise optimization and sensitivity analysis while minimizing computational demands. The design variables chosen for this study are the volute meridional inlet width, which ranges from 0.25 m to 0.3 m, and the Stepanoff number, which ranges from 0.3 to 0.5.

Response Surface Modeling (RSM) was subsequently employed to approximate the relationship between design variables and system performance. It provided a continuous and computationally efficient representation of the fan's behavior across the design space. RSM enables rapid evaluation of multiple design configurations, identification of trends, and sensitivity analysis by fitting a surrogate model to the CFD simulation results obtained from the DOE-LHS framework. This approach is widely used in engineering optimization where full-scale simulations or experiments are computationally expensive and time consuming.

Kriging response surface type, a powerful interpolation-based technique that combines regression modeling with spatial correlation between data points to provide highly accurate predictions even in complex and nonlinear design spaces is used in RSM in this study. Kriging is particularly effective for capturing localized variations and minimizing prediction errors by balancing global and local trends in the data. The model was applied to maximize the total efficiency of the centrifugal fan, enabling the identification of the most

effective combination of volute meridional inlet width and Stepanoff number without performing exhaustive simulations. To assess the predictive accuracy and reliability of the model, number of verification points was set to five.

The Multi-Objective Genetic Algorithm (MOGA) is an optimization technique that is based on the principles of the genetic algorithm, and it is used to solve problems with multiple objectives at the same time. The approach explores the design space via evolutionary operations such as selection, crossover and mutation. Candidate solutions are evaluated by their objective functions. MOGA does not seek a single solution but instead generates a set of non-dominated candidate solutions, each representing different trade-offs among the objectives, based on the concept of Pareto dominance [15]. After creating response surface MOGA method used to find out the optimum candidate points.

2.5. Fan Characteristic Curve

The relationship between flow rate and pressure rise is represented by a fan characteristic curve which provides important information about performance under various operating conditions. It helps to match the fan to the system resistance for optimal operation and represents the best efficiency point (BEP). In CFD studies, characteristic curves can be generated by simulating the fan at different outlet pressure conditions to determine flow rates, allowing detailed assessment of performance and efficiency [16]. In this study, fan characteristic curves were determined for both initial design and optimized model to compare between each other. The same methodology and set up used as in the mesh independence study. Outlet pressure set as input parameter in CFX, to evaluate total efficiency, volumetric flow rate, and other performance metrics across different pressure values.

3. RESULTS

3.1. Optimum Stepanoff Number and Meridional Inlet Width Values for the Optimized Design

After the study of mesh independence 3rd case which includes 2269760 number of mesh elements was chosen for optimization study. The results converged at 10^{-4} as residual target was set. Design of Experiments (DOE) was performed with Latin Hypercube Sampling Design in Ansys. Two design variables were selected as Stepanoff number and volute meridional inlet width. Nine design points were automatically generated in the given ranges of these design variables. CFD simulations were then carried out using Ansys CFX for each design point, and the results are shown in Table 6.

Table 6. Design of experiments (DOE) results

Effect of different volute meridional inlet width and Stepanoff number on fan performance			
Volute meridional inlet width (mm)	**Stepanoff number**	**Total efficiency (%)**	**Volume flowrate (m^3/s)**
275	0.333	33.77	2.884
252	0.311	34.66	2.904
258	0.466	24.08	2.649
263	0.422	27.58	2.728
291	0.355	32.43	2.849
280	0.4	29.22	2.766
297	0.444	25.72	2.695
269	0.488	22.90	2.617
286	0.377	31.09	2.807

Based on the results of CFD from generated design points, a response surface was constructed by using Kriging metamodel. Five verification points were generated automatically and two more added manually to verify the created metamodel. Design points that showed relatively large difference from the ideal prediction line were selected as refinement points to improve the accuracy of the metamodel. Result of predicted vs observed model is shown in Figure 4.

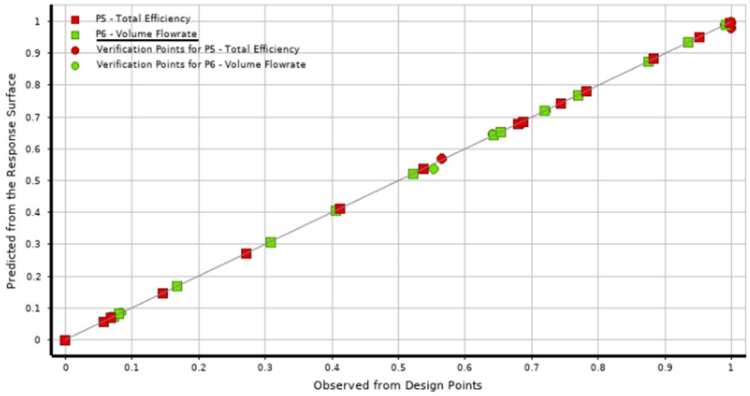


Figure 4. Predicted vs observed values of response surface

From the created metamodel, relative maximum absolute error was 4.86% for total efficiency and 4.19%, for volume flowrate while relative average absolute error was 1.28% for total efficiency and 0.93% for volume flowrate. The response result of metamodel and the relation in between volute meridional width, Stepanoff number and total efficiency is seen in Figure 5.

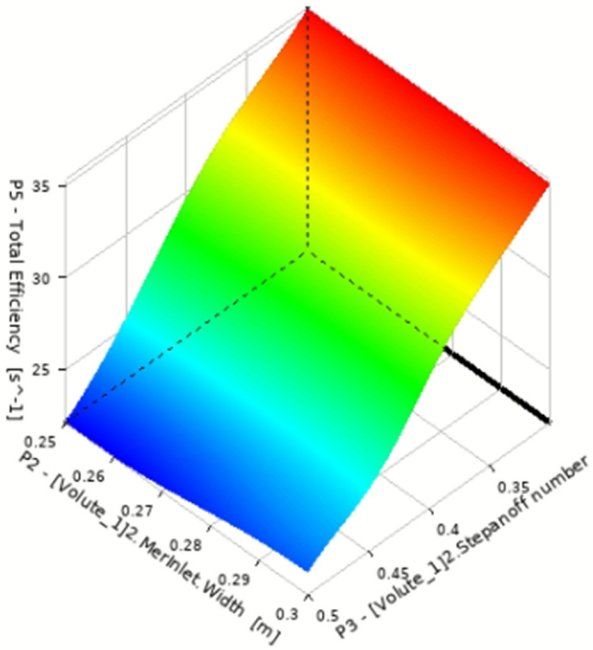


Figure 5. Effect of volute meridional inlet width and Stepanoff number on total efficiency

Optimized geometry candidates were found out by applying MOGA method. After 3755 evaluations, the optimization was converged. The pareto percentage is 0.25% and the stability percentage is 1.23%. Three candidates are shown in Table 7.

Table 7. Candidates for optimum model

Candidates	Volute meridional inlet width	Stepanoff number	Verified total efficiency (%)
First candidate	0.272 m	0.3	35.188
Second candidate	0.277 m	0.3	35.12
Third candidate	0.25 m	0.3	35

Response Surface study clearly shows that total efficiency is highest when the volute meridional inlet width is approximately 0.27 m and the Stepanoff number is around 0.3. Based on this analysis, a new fan volute was designed with a volute meridional inlet width of 0.27 m and a Stepanoff number of 0.3. Prototype and Optimized design are shown in Figure 6.

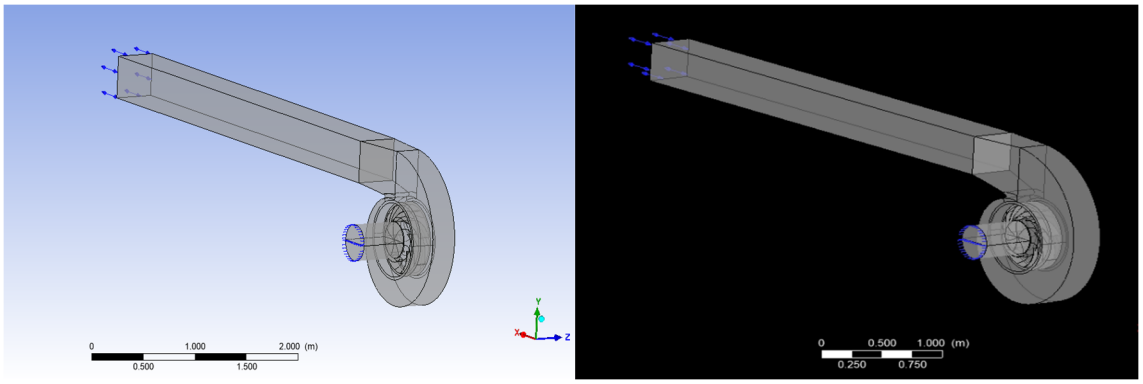


Figure 6. Prototype (left) vs optimized design (right)

3.2. Fan Characteristics of Prototype and Optimized Geometry

Both the prototype and optimized centrifugal fan geometries were analyzed using the same methodology to ensure methodological consistency. Outlet pressure was defined as the input parameter for analyzing total efficiency and volume flow rate under different operating conditions. Table 8 shows the values of volume flowrate and total efficiency of optimized and prototype geometry. In Table 8, the results indicate a notable improvement in total efficiency within the low and medium pressure ranges. Especially, in the pressure range of 0–900 Pa, corresponding to typical low-to-medium pressure conditions for this centrifugal fan, it was achieved a relative average efficiency gain of approximately 6.79%. This demonstrates that the optimized fan geometry provides superior performance in these operating ranges. Since the optimization mainly focuses on low-to-medium pressures, the optimized fan shows a slight decrease in efficiency when compared to the prototype at pressures above 900 Pa. Moreover, the optimized fan maintains slightly higher or comparable volume flow rates across most of the pressure spectrum, confirming that the redesign enhances aerodynamic performance without compromising airflow capacity.

Table 8. Comparison of the prototype and the optimized fan performance at different outlet pressure values

Pressure values (Pa)	Prototype fan		Optimized fan	
	Volume flowrate (m^3/s)	Total efficiency (%)	Volume flowrate (m^3/s)	Total efficiency (%)
0	2.781	29.72	2.912	35.1
100	2.698	39.11	2.846	45.63
200	2.613	47.15	2.722	51.637
300	2.527	54.08	2.620	57.992
400	2.437	60.05	2.514	63.298
500	2.343	65.45	2.405	67.811
600	2.233	69.51	2.290	71.591
700	2.117	72.82	2.166	74.618
800	1.998	75.77	2.033	76.877
900	1.870	77.92	1.887	78.265
1000	1.729	79.24	1.727	78.857
1100	1.569	79.71	1.529	78.148
1200	1.337	78.06	1.22	75.163
1300	0.117	20.19	0.06	12.12

4. CONCLUSIONS

The paper optimizes the efficiency of a centrifugal fan for silo aeration by selecting the volute meridional inlet width and Stepanoff number as design variables. The CFD analysis was carried out using Ansys CFX and DOE was employed to examine the effects of the chosen parameters on total efficiency and volume flow rate. A Kriging-based metamodel was used to determine the response surface and optimal design.

The optimization results showed that the optimal configuration was obtained at the volute meridional inlet width that is approximately 0.27 m and the Stepanoff number of approximately 0.3. The comparison

between the prototype and optimized fan demonstrated 6.79% relative average efficiency improvement in the 0–900 Pa pressure range. The improvement was more significant in low pressure operating conditions and the optimized fan maintained slightly higher volumetric flow rates over most of the investigated pressure range. The rate of improvement in efficiency gradually decreased above 900 Pa pressure and the optimized fan had similar or slightly lower efficiency than the prototype as pressure increased.

The results generally indicate that the optimization of the selected volute variables improved the efficiency of the centrifugal fan with a particular emphasis on the desired low- and medium-pressure operating range. The combined use of CFD, DOE and Kriging-based response surface optimization gave an effective approach for determination of the improved fan geometry while evaluating the effects of the design parameters on fan performance. Future study can be conducted by comparing the results with experimental fan models and more design variables can be considered to optimize other parts of the fan system.

5. REFERENCES

1. U.S. Department of Energy (2025). *Energy efficiency and renewable energy program.* Improving Fan System Performance - A Sourcebook for Industry. https://docs.nrel.gov/docs/fy03osti/29166.pdf, Access date:10.11.2025

2. Enerji ve Tabii Kaynaklar Bakanlığı (2017). *Endüstriyel sistemlerde optimizasyon-Fan sistemleri*. https://www.enerji.gov.tr, Access date: 08.11.2025

3. Meng, F., Dong, Q., Wang, Y., Wang, P. & Zhang, C. (2013). Numerical optimization of impeller for backward-curved centrifugal fan by response surface methodology. *Research Journal of Applied Sciences, Engineering and Technology, 6*(13), 2436-2442.

4. Wu, Y. & Huang, D. (2019). Optimization design of axial fan blade. *Journal of the Chinese Institute of Engineers, 42*(6), 473-478.

5. Jung, U.-H., Kim, J.-H., Kim, J.-H., Park, C.-H., Jun, S.-O. & Choi, Y.-S. (2016). Optimum design of diffuser in a small high-speed centrifugal fan using CFD & DOE. *Journal of Mechanical Science and Technology, 30*(3), 1171-1184.

6. Kim, J.H., Oh, K.T., Pyun, K.B., Kim, C.K., Choi, Y.S. & Yoon, J.Y. (2012). Design optimization of a centrifugal pump impeller and volute using CFD. *IOP Conf. Series: Earth and Environmental Science, 15*(3), 032025.

7. Liu, X., Dang, Q. & Xi, G. (2008). Performance improvement of centrifugal fan by using CFD. *Engineering Applications of Computational Fluid Mechanics, 2*(2), 130-140.

8. Ragauskas, P., Tetsmann, I. & Jasevičius, R. (2024). The optimization of the geometry of the centrifugal fan at different design points. *Applied Sciences, 14*(8), 3530.

9. Yaşar, A., Aygül, Y. ve Çelebi, K. (2024). Torsiyonel rijit fan kaplinlerinin tasarlanması ve optimizasyonu. *Çukurova Üniversitesi, Mühendislik Fakültesi Dergisi, 39*(4), 939-950.

10. *CFD software: Fluid dynamics simulation software.* ANSYS. https://www.ansys.com/products/fluids Access date: 16.10.2025.

11. Versteeg, H.K. & Malalasekera, W. (2007). *An introduction to computational fluid dynamics: The finite volume method.* 2nd edition. London. 503.

12. Jog, C.S. (2015). *Fluid mechanics: Foundations and applications of mechanics.* 3rd ed., Cambridge University Press.

13. *Ansys CFX reference guide.* ANSYS Inc., Canonsburg, PA. Access date: 06.11.2025

14. Lee, Y.-T., Ahuja, V., Hosangadi, A., Slipper, M.E., Mulvihill, L.P., Birkbeck, R. & Coleman, R.M. (2011). Impeller design of a centrifugal fan with blade optimization. *International Journal of Rotating Machinery, 2011*(1), 537824.

15. Fonseca, C.M. & Fleming, P.J. (1993). Genetic algorithms for multi objective optimization: Formulation, discussion and generalization. *Genetic Algorithms: Proceedings of the Fifth International Conference, San Mateo,* CA: Morgan Kaufmann, 416-423.

16. Ferguson, C.G. (1942). Fans-their types, characteristics, and application. *Proceedings of the Institution of Mechanical Engineers, 148*(1), 1-25.