

Original Article

Design, Development, and Computational Analysis of a Twin Lobe Rotor for Positive Displacement Blowers

Swati Pradip Shinde^{1*}, D.B Jadhav²

^{1,2}Depatmnet of Mechanical Engineering, Bharati Vidyapeeth (Deemed to be University) College of Engineering, Pune, Maharashtra, India.

^{1*}Corresponding Author: shindeswati2009@gmail.com

Received: 01 June 2026

Revised: 18 September 2026

Accepted: 04 october 2026

Published: 26 October 2026

Abstract - Twins lobe blowers are used for many industrial applications, such as wastewater treatment plants and pneumatic conveying and combustion air supply systems, but their efficiency is dependent on rotor designs. The research study evaluates the complete development process and operational testing procedure of a twin lobe blower, which uses new rotor designs to improve its compression performance, operational efficiency, and mechanical durability. A new rotor design, which uses involute curves as its basis, has been developed to replace existing conventional design techniques for 0.069 m³ capacity blowers. The design process starts with parametric modeling and regression analysis, which leads to three-dimensional solid modeling that creates a design that undergoes testing through computational simulations that assess airflow patterns, temperature patterns, pressure patterns, velocity patterns, and streamlines. The study shows that the modified rotor design performs better than the regular design because it maintains pressure better, decreases leakage, and provides better thermal performance, which results in higher overall efficiency. The results show that changes to the rotor profile create strong effects on flow parameters, which lead to inlet leakage and outlet backflow, and transient changes in velocity, mass flow rate, and exit temperature. The proposed method eliminates the requirement for reverse engineering and complex machining processes, which leads to simpler manufacturing operations and shorter production periods. The research establishes essential principles that enable the creation of twin lobe blowers that achieve high performance through their operational reliability and efficiency improvements.

Keywords - Twin lobe blower, Rotor, CFD analysis, Involute curves, Regression analysis, Pressure contours, Three-dimensional solid modeling.

1. Introduction

Twin lobe blowers/compressors have gained widespread industrial application because their operating efficiency and mechanical reliability exceed those of conventional reciprocating and turbo compressors at medium flow rates and power levels [1]. The fundamental design of these compressors enables them to raise the pressure of gas or vapor within a sealed system through positive-displacement compression [2]. Twin-Screw Compressors (TSCs) or blowers operate as positive displacement machines, and the industrial sector relies heavily on them for gas compression due to their high volumetric efficiency and robust performance across diverse operating conditions [3]. Within this machine class, rotary lobe blowers-commonly referred to as direct-lobed compressors or Roots blowers-constitute an essential category of rotary positive-displacement compressors [4]. The structural configuration of a typical lobe blower comprises two driving lobe rotors mounted on parallel shafts, a cylinder, two side end-plates, and a pair of synchronized timing gears [4]. Operating as positive displacement pumps, Roots blowers generate continuous airflow by trapping discrete volumes of

fluid within their rotating rotors [5]. Motion control is maintained via synchronized timing gears, enabling the bi-lobe rotors to rotate in opposite directions without physical contact [6]. The fluid compression sequence initiates when the rotating lobes isolate fluid from the inlet port within the housing cavity, transporting it toward the discharge port and downstream system [6].

In terms of rotor configuration, two-lobe Roots blowers are predominantly selected for applications requiring high flow rates, whereas three-lobe configurations are utilized to reduce backflow and suppress pressure pulsation effects [3]. Scientists have conducted research about Roots blowers, which they now understand through testing their standard rotor designs and computer simulations, because they face challenges in obtaining complete access to detailed geometric specifications and complete performance assessment techniques. The research system operating procedures face multiple restrictions because they cannot detect all system flow patterns that exist in the current methods to detect backflow and leakage, and measure temperature changes that



affect blower performance. The current design methods, which use empirical data, do not enable engineers to develop optimized rotor solutions that comply with present-day industrial standards. Researchers need to establish a better design method for creating rotor systems that require mathematical models and advanced computer simulation methods. The research presents a novel twin-lobe rotor design, which develops its design through parametric equations and regression methods and tests its performance through extensive CFD testing to achieve improved pressure performance, reduced leakage and higher operational efficiency. The key contributions are:

- The new twin-lobe rotor profile is developed through the use of parametric equations and regression-based modeling, which provides a precise geometric representation.
- Conducted a complete CFD analysis to assess the blower's pressure, velocity and temperature and internal flow patterns.
- The proposed rotor design improves performance by enhancing pressure distribution, reducing leakage losses and achieving better flow uniformity than the traditional design.
- The design framework provides a systematic approach that reduces dependence on empirical methods while enabling efficient development of manufacturable rotor designs.

The literature review in Section 2 examines current research while showing recent progress and existing gaps in research about twin-lobe Roots blower design and analysis. The proposed methodology in Section 3 shows rotor profile development through parametric equations and regression techniques, together with computational modeling methods. The CFD analysis in Section 4 shows its results through pressure, velocity and temperature contours and flow streamlines. Section 5 provides a comparative performance evaluation between the conventional and proposed rotor designs. The paper concludes in Section 6, which presents essential discoveries and reviews future research possibilities.

2. Literature Review

The Twin lobe Roots blowers supply continuous oil-free airflow, which makes them essential for industrial processes. The system performance depends on three variables, which include rotor design, airflow patterns and their operating conditions. The designers of traditional systems select standard rotor shapes, which include circular, involute and cycloidal designs that engineers developed through practical testing and reverse engineering techniques. The methods become ineffective when they attempt to enhance efficiency

while managing leakage and meeting current performance standards. This section reviews recent advancements in rotor design and flow analysis with optimization techniques that enhance twin lobe Roots blower performance and reliability.

Monazami et al. [4] developed an ultrasonic-based non-destructive technique that detects backflow in lobe blowers through flow irregularity and pressure variation analysis. The method introduces the crest factor ratio as an effective parameter for identifying abnormal flow behavior and validates the results experimentally. The approach provides detection capabilities, but it does not help to create advanced rotor designs or to boost operational effectiveness. The system needs specific testing locations together with ultrasonic testing instruments, which create obstacles for industrial facilities to use and reduce their operational range.

Bikramaditya et al [9] created a CFD-based shape optimization system that used an adjoint-sculpting approach to design Roots blower rotors. The approach involves the use of RBF morphing and adjoint solver functions to design precise rotor tip shapes that do not require constant remeshing. The system provides improved designs as it uses less computing power than conventional trial-and-error techniques. The method is difficult to use because it requires expertise to perform the complex calculations, which makes it unrepresentative in industrial applications. The system is mainly concerned with improving tip performance and does not offer solutions for the design of the rotor or direct experimentation.

Tran and Nguyen [10] developed a mathematical and Computational Fluid Dynamics (CFD) modelling system to examine the flow patterns in a cylindrical and helical multi-lobe Roots blower. The authors tested pressure and flow characteristics to analyse the efficiency of helical lobes and demonstrated their ability to provide uniform flow patterns with minimal fluctuations. The team discovered that helical lobes offer better pressure and reduced vortex formation. The research focuses on specific rotor designs, but the study does not consider novel and hybrid rotor designs. This study uses simulations as the main testing method since it contains a limited number of experimental tests, which limits its ability to operate in practice.

Bikramaditya et al. [11] also created a Computational Fluid Dynamics (CFD) model to study the radial leakage flow of a Roots blower, and they validated their model by using Particle Image Velocimetry (PIV) experiments. The researchers created sophisticated grid generation techniques that produced 2D and 3D deforming mesh models that reached high precision since their velocity measurements showed deviations within $\pm 5\%$ of experimental results. The system successfully simulated all aspects of leakage flow patterns and temperature distribution, plus discharge behavior that occurred during different testing scenarios. The research

mainly studies leakage flow patterns instead of developing better rotor designs, which would increase overall system efficiency. The model needs high processing power because it uses simulation tests to function, which makes it impossible to operate in industrial environments.

Ubiola-Soto et al. [12] introduced Surface Response Design of Experiments (SRDOE) as a methodology to examine how variations in pad clearance and preload will impact both system stability and critical speed, together with vibration performance. The research shows that non-uniform pad designs create higher effects on amplification factors and stability margins than uniform tolerance assumptions. The study investigates bearing dynamics in turbomachinery systems, but it does not analyze either the aerodynamic flow patterns or the rotor profile design. The study investigates mechanical stability, but it does not investigate fluid flow characteristics and leakage effects that impact twin-lobe Roots blower performance.

Bikramaditya et al. [13] developed a numerical method that includes coupled solvers and dynamic mesh techniques to study thermal deformation in Roots blowers. The system showed how temperature expansion changes operational gaps, which affects both leakage rates and system performance. The model depends on simulation assumptions, which restrict its ability to replicate actual experimental environments. The system requires extensive computational power and resources, which makes it difficult to use for quick design improvements.

Enlong et al. [14] investigated how a viscous material moves through a two-lobe rotary pump by using dynamic mesh simulation together with micro-filling experimental research. The researchers adopt a Computational Fluid Dynamics (CFD) dynamic mesh model, which enables them to track the changes in flow patterns as they compare their results with experiments. The research demonstrates that the rotor motion, clearance sizes and the filling of the cavities affect the flow of viscous materials. The study is limited to liquid transport for the viscous fluid and does not include any of the gas flow characteristics of blower systems, which are compressible. This study fails to investigate rotor profile optimization and aerodynamic performance, which are important aspects in the design of twin-lobe Roots blowers.

Ma et al. [15] created a comprehensive approach using thermodynamic modeling coupled with exergy analysis and Computational Fluid Dynamics (CFD) simulation to study leakage losses, gas forces and system efficiency. The research identified that the interloop leakage acts as the major source of power loss, and system efficiency rises with increasing speed as a non-linear curve, which generates lesser benefits at very high speeds. The operating system of the device needs the lobe number and wrap angle to set up pressure variations and volumetric efficiency. The research work examined only the screw compressor system, as it does not include twin-lobe

Roots blower systems and rotor profile optimization for blowers. The study emphasized thermodynamic analysis and leakage evaluation, but it failed to deliver a complete examination of aerodynamic flow patterns together with experimental confirmation.

Lisowski and Augustyn [16] built a fundamental analytical framework, which they validated through wind tunnel tests and CFD simulations that enabled them to forecast torque and power with 1 percent accuracy. The study showed that the carousel rotor design produces better starting torque than conventional rotor designs, which helps machines to start automatically. The research work only investigates wind energy applications while studying how turbine blades perform aerodynamically. The system does not provide information about the design of rotor profiles or the behavior of leakage and internal flow patterns, which are essential for twin-lobe Roots blower systems.

Valand et al. [17] performed a study on twin-screw compressors, which used CFD simulation to demonstrate how different tooth profile geometries impacted leakage performance. The authors established multiple rotor tooth profiles through solid modeling software, and they assessed performance under different operational conditions, which included multiple clearance, pressure, velocity and temperature settings. The research found that internal leakage constitutes a primary factor that determines compressor efficiency, and the tooth profile and clearance optimization method achieves higher efficiency because it decreases operational losses. The research demonstrated that CFD exists as a cost-effective solution for compressor performance evaluation, which replaces traditional methods that require experimental testing. The research investigates only screw compressor systems, while it excludes twin-lobe Roots blowers and blower rotor profile design, and it lacks experimental tests, which should have been conducted under actual working conditions.

Li Zhang, Yang Liu et.al. [18] A novel smooth rotor profile consisting of eight segments of curves, including arcs and conjugate correction curves, is proposed for higher pumping speeds. Ngoc-Tien Tran et al. [10] designed a rotor for a tri-lobe Roots blower with dedendum a circular arc, the addendum profile having a partial arc, and an epicycloid. Z F Huang and Z X Liu [19] the evolutionary characteristics of flow for the positive displacement blower were identified and visually analysed by performing numerical simulation of unsteady compressible flow in an involute-type three-lobe positive discharge blower using computational fluid dynamics.

Antonio Giuffrida [20], A geometric study of different curves in this paper presents the results of comparisons among some patented solutions for profiling the contours of the rotors in twin-screw compressors. Referring to a base case where all

the generating curves are circumferences, patents suggesting to replace arcs of circumference with arcs of conic sections, i.e. parabola, ellipse and hyperbola, Involute profile curves have long been utilized in rotary positive displacement machinery due to their conjugate meshing characteristics, constant pressure angles, and uniform torque transmission capabilities. In screw compressor and rotary blower applications, involute-based designs offer favorable contact lines that help seal internal compression chambers. Patil and Abdan et al. demonstrated that modifying involute profiles through smooth arc transitions significantly impacts sealing efficiency and fluid friction losses along the rotor contact boundary. Similarly, Horst explored high-flow blower profiles to optimize non-contacting meshing zones, demonstrating that precise geometric parameterization of the profile flank directly mitigates inter-lobe clearance gaps. Despite these advantages, classical involute profiles often suffer from sharp transition points near the root and tip radii, leading to localized recirculation and manufacturing complexities if not combined with higher-order smoothing functions.

To capture the transient, multi-dimensional flow phenomena within rotary blowers, Computational Fluid Dynamics (CFD) combined with dynamic meshing techniques has become the standard design optimization tool. Bikramaditya et al. established a CFD-based shape optimization system using an adjoint-sculpting approach and RBF morphing to refine rotor tip shapes without constant remeshing, successfully reducing parasitic drag and pressure losses. In follow-up studies, Bikramaditya et al. validated radial leakage flow and thermal deformation models against Particle Image Velocimetry (PIV) experiments, proving that transient mesh deformations are essential for predicting micro-clearance leakage within $\pm 5\%$ accuracy.

2.1. Research Gaps

While computational and numerical methods have advanced the study of rotary machinery, critical gaps remain in the existing literature regarding twin-lobe blower rotor design:

- **Lack of Direct Mathematical Profile Generation:** Traditional empirical design methods do not provide engineers with a direct, flexible mathematical framework to construct optimized, non-standard rotor profile geometries that mitigate tip leakage without relying on reverse engineering or physical prototypes.
- **Absence of Integrated High-Order Curve Fitting with CFD Domain Setup:** Existing research rarely couples piecewise parametric equations with higher-order regression modeling (e.g., cubic polynomial fitting) to eliminate artificial boundary discretization errors and smooth out micro-clearance meshing zones in Computational Fluid Dynamics (CFD) models.
- **Insufficient Comparative Micro-Scale Flow Evaluation:** There is a lack of comprehensive transient CFD evaluations comparing conventional master rotor designs

against newly developed involute-arc profiles to quantify differences in pressure retention, velocity gradients, thermal buildup, and recirculation streamlines under identical operating conditions.

2.2. Problem statement

Despite their broad application, the overall compression efficiency, volumetric performance, and mechanical durability of twin-lobe Roots blowers are heavily constrained by internal leakage losses and suboptimal aerodynamic profiles [4, 7]. Because rotary lobe blowers rely on non-contacting rotating components, internal clearance gaps—specifically radial gaps between the rotor tip and casing, inter-lobe meshing gaps, and axial end-plate gaps—are unavoidable [7].

The radial leakage gap, in particular, acts as the primary pathway for high-pressure fluid to backflow into lower-pressure intake chambers, causing significant parasitic energy losses and volumetric degradation [7]. Traditional design methodologies rely heavily on empirical formulas, standard fixed curves (such as simple circular, cycloidal, or standard involute shapes), and reverse engineering of existing rotor physical samples.

These conventional techniques are incapable of systematically optimizing the complex boundary geometries needed to minimize clearance leakage, suppress backflow, and limit excessive thermal buildup. Consequently, reliance on trial-and-error design cycles leads to inefficient compression performance and requires complex multi-axis machining processes.

3. Methodology

To bridge these gaps, this study presents a novel twin-lobe rotor design for a 0.069 m^3 capacity positive displacement blower. The design replaces conventional empirical methods with a rigorous mathematical formulation using parametric equations (combining involute curves and circular arcs) and regression analysis, followed by full 3D transient CFD simulations using the Finite Volume Method (FVM) to assess airflow patterns, pressure contours, velocity gradients, and thermal behavior. The method involves the need for researchers to generate and solve parametric equations to give the X and Y coordinates for plotting the rotor profile.

The generated coordinates undergo regression analysis, which results in an optimized representation of the data. The results create a three-dimensional solid model, which scientists test through computational simulation that employs the finite volume method. The simulation tests how air flows through the system while measuring temperature levels and tracking velocity patterns across different design modifications. The existing model and the newly developed rotor design are compared through graphical analysis of pressure, temperature, and velocity contours, which researchers extracted from the two designs.

3.1. Design of Twin Lobe Profile using Mathematical Equation

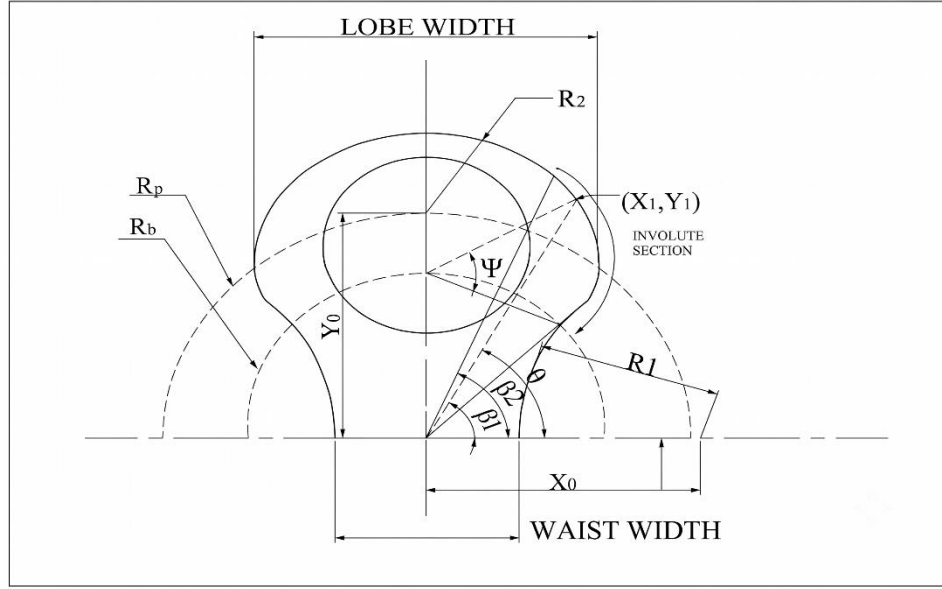


Fig. 1 Typical sketch of rotor profile

The objective of the present dissertation work is to design a new twin lobe profile through the process of solving governing equations, which will create geometries that serve as the foundation for upcoming simulation studies. The basic geometry considered here is of a circular involute and a circular arc.

Here,

X_0, Y_0 - X and Y co-ordinates from origin, θ -Angle specifying the polar coordinate system for the concerned curve, β_1 -Angle specifying the orientation of the first circular arc, starting from zero, i.e., the lower limit, β_2 -Angle specifying the orientation of the first circular arc, upper limit, R_1 -Radius of the lower circular arc, R_2 -Radius of the upper circular arc, R_b -Radius of the base circle, R_p -Radius of the pitch circle, Ψ -The angle of the involute with the base circle, K_{21}, K_{22} -The constants whose values depend on the value of Ψ .

Figure 1 below represents the rotor profile design.

Curve 1: If $\theta = [0, \beta_1]$

$$X(\theta) = R_1 \cos\left(\pi - \left(5\frac{\theta}{4}\right)\right) + X_0 \quad (1)$$

$$Y(\theta) = R_1 \sin\left(\pi - \left(5\frac{\theta}{4}\right)\right) \quad (2)$$

Curve 2: If $\theta = [\beta_1, \beta_2]$

$$X(\theta) = K_{21}(\Psi) \cos(-\beta_1) + K_{22}(\Psi) \sin(-\beta_1) \quad (3)$$

$$Y(\theta) = K_{22}(\Psi) \cos(-\beta_1) + K_{21}(\Psi) \sin(-\beta_1) \quad (4)$$

Here;

$$K_{21}(\Psi) = R_b (\cos(\Psi) + \Psi \sin(\Psi)) \quad (5)$$

$$K_{22}(\Psi) = R_b (\cos(\Psi) + \Psi \sin(\Psi)) \quad (6)$$

$$\theta = \tan^{-1} \left(\frac{\sin(\Psi) - \Psi \cos(\Psi)}{\cos(\Psi) + \Psi \sin(\Psi)} \right) \quad (7)$$

Curve 3: If $\theta = [\beta_2, \pi/2]$,

$$X(\theta) = R_2 \cos\left(\frac{52.6\pi(\theta-\beta_2)}{180\left(\frac{\pi}{2}-\beta_2\right)} + \frac{37.47\pi}{180}\right) \quad (8)$$

$$Y(\theta) = R_2 \sin\left(\frac{52.6\pi(\theta-\beta_2)}{180\left(\frac{\pi}{2}-\beta_2\right)} + \frac{37.47\pi}{180}\right) + Y_0 \quad (9)$$

Case I (a) Sample calculations

For $\theta = [0, \beta]$, Basic Data $R_p=58.674$, $R_b=0.8R_p$, $R=35.24$, $1^\circ=0.01745c$.

Table 1 and 3 below shows the sample calculations and summarized coordinates.

Table 1. Sample calculations

θ	0	0.25	0.5	0.75	1	1.25
R1	35.24	35.24	35.24	35.24	35.24	35.24
$\cos\{\pi-(5x\theta)/4\}$	-0.999999995	-0.9999857	-0.9999416	-0.9999	-0.9998	-0.9996

$\sin\{\Pi-(5x_0)/4\}$	-9.73464E-05	0.00535575	0.01080869	0.01626	0.02171	0.02716
x1	23.835					
x0	59.07499983	59.075	59.075	59.075	59.075	59.075
Now						
x2 to 40 (first curve x co-ordinates)	23.835	23.8355054	23.8370586	23.8397	23.8433	23.848
y1(first curve y co-ordinates)	-0.00343048	0.18873674	0.38089835	0.57305	0.76518	0.95729
For $\theta = [\beta_1, \beta_2]$		$[40^\circ, 70^\circ]$				
$\theta =$	tan-1	$[\sin\psi - \psi \cdot \cos\psi]$				
		$[\cos\psi + \psi \cdot \sin\psi]$				
Ψ	1	2	3	4	5	6
$[\sin\psi - \psi \cdot \cos\psi]$	1.77114E-06	1.4168E-05	4.7809E-05	0.00011	0.00022	0.00038
$[\cos\psi + \psi \cdot \sin\psi]$	1.00015224	1.00060882	1.00136932	1.00243	1.0038	1.00547
$\tan([\sin\psi - \psi \cdot \cos\psi] / [\cos\psi + \psi \cdot \sin\psi])$	1.77087E-06	1.4159E-05	4.7744E-05	0.00011	0.00022	0.00038
inverse tangent	0.000101463	0.00080214	0.0027502	0.0063	0.01261	0.02177
θ (addition of 40°)	40.00010146	40.0008021	40.0027502	40.0063	40.0126	40.0218
$K_{21}\psi = R_b \cdot [\cos\psi + \psi \cdot \sin\psi]$	46.94634601	46.9677775	47.0034749	47.0534	47.1175	47.1958
$K_{22}\psi = R_b \cdot [\sin\psi - \psi \cdot \cos\psi]$	8.31357E-05	0.00066502	0.00224412	0.00532	0.01038	0.01794
$\cos(-\beta_1)$	0.766127954	0.7661201	0.76609825	0.76606	0.76599	0.76588
$\sin(-\beta_1)$	-0.64268807	-0.6426974	-0.6427235	-0.6428	-0.6429	-0.643
$x_2(40, 70)$	$K_{21}\psi \times \cos(-\beta_1) + K_{22}\psi \times \sin(-\beta_1)$					
$y_2(40, 70)$	$K_{22}\psi \times \cos(-\beta_1) - K_{21}\psi \times \sin(-\beta_1)$					
$x_2(40, 70)$ --- (second curve x co-ordinates)	35.96690802	35.9829582	36.0092798	36.0457	36.0914	36.1465
$y_2(40, 70)$ ---- (second curve y co-ordinates)	30.17192029	30.1865798	30.2119563	30.2486	30.2977	30.3596
$y_3 = R_p + R_2$						
92.865	58.674 + R_2					
$R_2 =$	34.191					
For $\theta = [\beta_2, \Pi/2]$		$[70^\circ, 90^\circ]$				
	70	71	72	73	74	75
$52.6(\theta - \beta_2)$	0	53.23383	105.8338	158.4338	211.0338	263.6338
$(\Pi/2 - \beta_2)$	20	19	18	17	16	15
	37.4	37.4	37.4	37.4	37.4	37.4
$y_0 = R_p$	58.674	58.674	58.674	58.674	58.674	58.674
A =	37.4	40.20178	43.27966	46.71964	50.58961	54.97559
A = in rad	0.6526	0.7015	0.7552	0.8153	0.8828	0.9593
Cos A	0.7945	0.7639	0.7281	0.6857	0.6350	0.5741
Sin A	0.6073	0.6454	0.6855	0.7279	0.7725	0.8188
R_2	34.191	34.191	34.191	34.191	34.191	34.191
$x_3 = R_2 \times \cos A$						
$R_2 \times \cos A$ --- (Third curve x co-ordinate)	27.1644	26.1172	24.8949	23.4441	21.7113	19.6282
$R_2 \times \sin A$	20.7634	22.0662	23.4364	24.8877	26.4130	27.9957
$y_3 = R_2 \times \sin A + y_0$ ----- (Third curve y co-ordinate)	79.4374	80.7402	82.1104	83.5617	85.0870	86.6697

Table 1 shows how to create a twin lobe rotor profile through parametric equations that operate in three separate sections. The first region ($\theta=0$ to β_1) starts its base lobe curvature through trigonometric functions, which establish initial x and y coordinates based on a permanent circular path.

The second region (β_1 to β_2) applies involute curve relations, which use K_{21} and K_{22} as parameters to achieve profile smoothness and continuous transitions. The third region (β_2 to $\pi/2$) uses circular arc equations to create the lobe tip, which shows x values that decrease while y values increase,

resulting in a rounded shape. The combination of these three curve segments creates a rotor profile that stays continuous and smooth, which a machine requires for precise geometric

representation and blower applications to analyze airflow and operate computationally.

Case I (b) Summarized Total Co-Ordinates

Table 2. Summarized total co-ordinates

X	Y	X	Y	X	Y	X	Y	X	Y
23.8355	0.0000	25.1359	9.4865	28.8450	18.1110	34.6997	25.4500	.	.
23.8371	0.3809	25.1881	9.6714	28.9442	18.2755	34.9787	25.5826	.	.
23.8397	0.5730	25.2414	9.8561	29.0444	18.4396	35.1193	25.8454	.	.
23.8433	0.7652	25.2956	10.0404	29.1454	18.6031	35.2606	25.9756	.	.
23.8480	0.9573	25.3509	10.2245	29.2472	18.7660	35.4026	26.1051		
23.8537	1.1494	25.4071	10.4082	29.3500	18.9284	35.5453	26.2338		
23.8605	1.3414	25.4644	10.5917	29.4537	19.0906	35.6887	26.3617		
23.8684	1.5334	25.5226	10.7748	29.5582	19.2514	35.8328	26.4888		
23.8773	1.7254	25.5819	10.9576	29.6636	19.4121	35.9776	26.6152		
23.8872	1.9173	25.6421	11.1401	29.7699	19.5722	36.1231	26.7407		
23.8982	2.1092	25.7034	11.3222	29.8771	19.7317				
23.9102	2.3009	25.7656	11.5040	29.9851	19.8906				
23.9233	2.4927	25.8288	11.6855	30.0940	20.0490				
23.9374	2.6843	25.8931	11.8666	30.2038	20.2067				
23.9525	2.8759	25.9583	12.0474	30.3144	20.3638			25.2469	63.7364
23.9687	3.0674	26.0245	12.2278	30.4259	20.5204			24.2625	64.7579
23.9860	3.2588	26.0916	12.4078	30.5382	20.6763			23.2317	65.7754
24.0043	3.4501	26.1598	12.5875	30.6514	20.8316			22.1540	66.7871
24.0236	3.6413	26.2289	12.7668	30.7654	20.9863			21.0285	67.7914
24.0440	3.8323	26.2990	12.9457	30.8803	21.1403			19.6282	68.6697
24.0654	4.0233	26.3701	13.1243	30.9960	21.2938			17.1075	69.5474
24.0879	4.2142	26.4421	13.3024	31.1125	21.4466	.	.	14.0378	70.4251
24.1114	4.4049	26.5152	13.4802	31.2299	21.5987	.	.	10.2781	71.3028
24.1359	4.5955	26.5892	13.6575	31.3481	21.7503	.	.	5.6556	72.1805
24.1615	4.7859	26.6641	13.8345	31.4671	21.9011	.	.	0.0000	73.0582
24.1881	4.9763	26.7400	14.0110	31.5869	22.0514	.	.		
24.2158	5.1664	26.8169	14.1871	31.7076	22.2009	.	.		
24.2445	5.3564	26.8948	14.3628	31.8290	22.3498	.	.		
24.2742	5.5463	26.9736	14.5381	31.9513	22.4981	.	.		
24.3050	5.7360	27.0533	14.7129	32.0744	22.6457	.	.		
24.3367	5.9255	27.1340	14.8873	32.1983	22.7926	.	.		
24.3696	6.1148	27.2157	15.0613	32.3230	22.9388	.	.		
24.4034	6.3040	27.2983	15.2348	32.4485	23.0843				
24.4383	6.4930	27.3818	15.4078	32.5747	23.2292				
24.4743	6.6818	27.4663	15.5804	32.7018	23.3733				
24.5112	6.8703	27.5518	15.7526	32.8297	23.5168				
24.5492	7.0587	27.6381	15.9242	32.9583	23.6596				
24.5882	7.2469	27.7254	16.0954	33.0877	23.8016				
24.6282	7.4348	27.8137	16.2661	33.2179	23.9430				
24.6693	7.6226	27.9028	16.4364	33.3488	24.0836				
24.7114	7.8101	27.9929	16.6061	33.4805	24.2236				
24.7545	7.9973	28.0840	16.7754	33.6130	24.3628				
24.7986	8.1844	28.1759	16.9441	33.7462	24.5013	36.6829	43.9444		
24.8437	8.3712	28.2688	17.1124	33.8802	24.6390	36.4684	44.7341		

24.8899	8.5577	28.3625	17.2801	34.0150	24.7760	36.2278	45.5446		
24.9370	8.7440	28.4572	17.4473	34.1504	24.9123	35.9600	46.3754		
24.9852	8.9300	28.5528	17.6140	34.2867	25.0479	35.6638	47.2256		
25.0344	9.1158	28.6493	17.7802	34.4236	25.1827	35.3384	48.0944		
25.0847	9.3013	28.7467	17.9458	34.5613	25.3167	34.9823	48.9813		

Table 2 shows the complete geometric attributes of the twin lobe rotor through its parametric and regression-based modeling. The data shows a gradual and continuous increase in both X and Y values, which creates a smooth and continuous path that extends from the base of the lobe to its tip. The profile begins with a slow height increase, which moves upward through width as it shows the root section of the profile, but the mid-region grows through a consistent linear development. The higher coordinate values make the curvature pattern more visible because it displays the rounded lobe tip and its surrounding transitional areas. The higher Y and X values in subsequent sections show that the rotor geometry reaches its full profile and maintains a symmetrical design. The existing minor shape changes and the non-linear behavior of the design require higher-order functions as regression models because they need to achieve complete shape accuracy. The dataset establishes a continuous rotational profile that manufacturing processes can create while performing CFD analysis and blower operations.

3.2. Regression Analysis

Correlation and Regression analysis are used as their primary techniques to examine the relationships between different variables. One measurement technique shows the strength of two variable relationships, while the other technique predicts how one variable will change when the second variable experiences different values.

3.2.1. Correlation Analysis

The Pearson correlation coefficient (r) assesses the strength and direction of an association between two variables.

Correlation Coefficient (r): 0.842

Strength: The measurements of lobe width and height show a strong positive relationship because both measurements depend on one another. The height (Y) of an object increases when its width (X) expands beyond its current measurement. The higher value demonstrates continuous geometric growth, which creates a nearly linear relationship with curved lobe profiles.

Simple Linear Regression

Therefore, the simple regression equation for modeling the relationship is:

Regression Equation:

$$Y = 4.38X - 105.14$$

Slope (m): 4.38 - For every 1 unit increase in width, the height increases by approximately 4.38 units.

Coefficient of Determination (R^2): 0.709.

The height variation is explained through 70.9% of its variance by the measurement of width. The remaining 29.1% of the variance happens because the lobe profile displays its natural curvature that a straight-line measurement cannot entirely capture.

3.2.2. Practical Observations for Blower Profile

The twin-lobe blower system provides detailed engineering details through a coordinate system.

Non-Linearity

The profile shows complex curvatures because the R^2 value shows “unexplained” variance, which remains in the presence of linear correlations. The results indicate that the linear regression sets the general trends, while the polynomial regression using second- and third-degree functions yields better results for manufacturing and Computational Fluid Dynamics (CFD) testing.

Coordinate Range

The data provides a specific region of operations through the coordinate range. The high slope of 4.38 reveals the rapid change of height relative to width in the tip zone of a lobe when these coordinates are used to establish the “meshing” area. The findings show that the linear model is better than the polynomial regression model as it has a smaller coefficient of determination.

3.2.3. Second-Order Polynomial Regression

By fitting the data to a quadratic equation, the curvature of the lobe is accounted for:

Polynomial Equation:

$$Y = -0.0987 X^2 + 10.151X - 188.42$$

Coefficient of Determination (R^2): 0.814

The linear model explained 70.9% of the variance. The polynomial model explains 81.4%. The width and height relationship shows a curved pattern because the R^2 value increased from a linear relationship.

3.2.4. Detailed Correlation Breakdown

The Pearson correlation coefficient $r = 0.842$ demonstrates a strong positive relationship between the two variables. The residuals exist as actual Y values minus predicted Y values, which show how much the model fails to match the real data. Table 3 shows actual height and predicted height values for each width measurement together with their residual errors.

Table 3. Sample residual estimation

Width (X)	Actual Height (Y)	Predicted Height	Residual (Error)
23.83	0.00	0.42	-0.42
25.13	9.48	8.95	0.53
28.84	18.11	18.45	-0.34
31.42	24.08	23.98	0.10
34.69	25.45	25.12	0.33
37.18	41.70	41.92	-0.22
34.59	49.88	49.65	0.23

The predicted values closely match the actual measurements, which demonstrates that the regression model effectively models the data. The residuals show small values that alternate between positive and negative results without showing any pattern of overestimation or underestimation. The data shows two different patterns because it underpredicts at lower widths, while it overpredicts at certain mid-range widths. The small magnitude of errors indicates the model is accurate and can be used to predict results. The regression model is able to define the relationship between width and height for the rotor profile according to the findings.

- The linear model overestimates height when the width is less than 26. The polynomial model fits the Flare of the lobe base better.

- The Mid-Range (28 - 33) is the most stable part of the lobe because the two models perform their role well.
- The curve begins to flatten out towards the lobe tip because upper width values are more than 35. The negative X2 coefficient (-0.0987) of the polynomial equation is correct because the curve starts to round off at the lobe tip.

3.2.5. Statistical Breakdown of the 3rd-Order Model

The cubic model demonstrates superior accuracy compared to all other models, which evaluate twin-lobe coordinates through an R2 value measurement that reached 0.912. The two coordinate sets show a strong correlation because their new coordinates create a geometric pattern that follows specific mathematical rules. The 3rd-order term (0.005X3) enables us to observe how the profile's curvature changes from its base root to its tip. The equation accounts for more than 91% of the variance, which enables Ansys to create smooth intermediate points that improve mesh quality in micro-clearance zones. The 3rd-order polynomial defines the boundary wall, which helps with sensitivity analysis because it produces a mathematically optimized curve that calculates mass flow leakage rates for CFD. The method decreases artificial leakage, which occurs because of discretization when a linear profile is poorly fitted.

Table 4. Comparison between linear, quadratic and cubic models

Model Type	Equation	R ² Score	Accuracy Interpretation
Linear	$Y = 4.38X - 105.14$	0.709	Captures the general upward trend but misses the profile's curve.
Polynomial (2nd)	$Y = -0.098X^2 + 10.15X - 188.4$	0.814	Accounts for the primary curvature of the lobe.
Polynomial (3rd)	$Y = 0.005X^3 - 0.58X^2 + 25.1X - 338.2$	0.912	Best Fit. Captures the complex inflection points of the rotor.

In Table 4, the linear model has a moderate fit ($R^2 = 0.709$), which demonstrates that the model can only detect the general upward trend, while it cannot show the profile's curved movements. The second-order polynomial has a better fit ($R^2 = 0.814$) as it uses primary curvature to represent the lobe shape.

The third-order polynomial has the best fit ($R^2 = 0.912$), which demonstrates the ability to capture the fluctuations and flexes in the rotor profile. The study establishes that model accuracy advances through increased complexity, which results in the cubic model delivering the most precise estimation.

3.2.6. Regression Analysis Summary

Table 5. Summary of regression analysis

Metric	Value	Technical Significance
Correlation (r)	0.842	Strong linear relationship between width and height
Cubic Equation	$Y = 0.005X^3 - 0.58X^2 + 25.1X - 338.2$	Best fit for non-linear rotor lobe curves
R-Squared (R ²)	0.912	91.2% of the geometry is explained by this model
Standard Error	1.84	Indicates high precision for coordinate prediction

Table 5 is the summary of the above regression analysis. The correlation coefficient ($r = 0.842$) establishes a strong positive linear relationship between width and height because both measurements show a consistent geometric relationship. The cubic equation provides the best representation of the rotor lobe profile, which shows its non-linear curvature and inflection points. The R^2 value of 0.912 demonstrates that

91.2% of the geometric variation is explained by the model, which confirms the model's high overall goodness of fit.

The standard error of 1.84 demonstrates high precision for predicting coordinate values because the measurements show only minimal deviation from actual values.

3.2.7. Regression Outcome

The correlation and regression analysis conducted on the twin-lobe blower profile coordinates shows a strong functional relationship between rotor lobe width (X) and its height (Y). The 3rd-order polynomial regression model showed statistical significance through its F-statistic, which reached high values while its p-value remained below the standard $\alpha = 0.05$ threshold. The Coefficient of Determination (R^2) shows that the cubic model explains almost all the profile geometry variations while it produces coordinates that precisely match the desired fluid-dynamic characteristics. The Root Mean Square Error (RMSE) results demonstrate that the mathematical model describes the physical rotor with high precision, which is essential for maintaining stable micro-clearances throughout production and assembly processes. The statistical validation results supply sufficient confidence to continue with the high-precision machining of the blower components.

3.3. Technical Implications for Rotor Design

The graph slope establishes a situation that causes even small changes in lobe width (X) to produce significant variations in height (Y) measurement. The height measurement serves as an essential factor for volumetric efficiency calculations because it determines the displacement volume that occurs during each engine rotation. The CNC program will create “steps” in the profile according to the polynomial residuals that show these coordinates as their current implementation. The quadratic fit will provide a better surface finish because it decreases fluid flow turbulence through its smoother results. The study developed the models, which are based on 3rd-order (Cubic) regression models, to create a precise statistical analysis of the rotor lobe profile. Engineering applications that require high precision work need this level of detail to define meshing geometry in patent applications and establish CFD domain boundaries.

The key benefits of regression analysis and simulations are-

- **CFD Mesh Boundary Quality:** In ANSYS, defining the boundary wall with a third-order polynomial allows the software to generate smooth intermediate boundary mesh nodes. This eliminates artificial mass flow leakage caused by discretized “jagged” profiles in micro-clearance gap simulations.
- **Machining Surface Finish:** Using polynomial residuals directly in CNC G-code programming prevents artificial “stepping” on the rotor surface. When machining complex non-circular curves like lobe rotors, standard CNC controllers process toolpaths using discrete linear segments (G01) or simple arc interpolations (G02/G03). Because lobe rotors feature continuously changing radii of curvature, fixed-step G-code outputs cause “stepping” (faceting or chordal deviation artifacts) on the finished rotor surface. A smooth polynomial profile reduces fluid turbulence across the inter-lobe and tip clearances.

Utilizing regression analysis residues in Computer-Aided Manufacturing (CAM) and toolpath generation algorithms systematically eliminates this stepping effect through mathematical optimization.

- **Volumetric Efficiency Optimization:** The tight standard error (1.84) guarantees that the theoretical displacement volume is accurately represented without unintended clearance variations during rotation.
- **Elimination of Empirical Methods:** Establishing a verified statistical profile replaces traditional reverse-engineering and empirical trial-and-error methods, shortening the design and production cycle.

3.4. Design Calculations

The parametric equations need to be solved first before determining the abscissa and ordinate of the solution. The relationship between these coordinates can be found by regression analysis. The figures below present the new rotor design together with its isometric representation in Figure 2.



New Rotor

Isometric Image

Fig. 2 New rotor and isometric image

3.4.1. Tools Used for Model Design and Analysis

The research method for this study includes two different phases that require design work and analysis work. The design phase created a three-dimensional model through the use of three-dimensional modeling software. The analysis phase used ANSYS to perform computational simulations, which utilized the Finite Volume Method.

4. Analysis and Results

The analysis was conducted through ANSYS software, during which pressure contours, velocity contours, temperature distributions, and surface streamlines were assessed for both the master rotor profile and the updated rotor profile. The results from both models were compared to determine the most efficient configuration.

4.1. CASE-I-Mesh

CASE-I As used in a computational scientific study, the term “mesh” primarily means to refer to the first mesh set up as the reference configuration. Figure 3 shows the details of the Mesh.

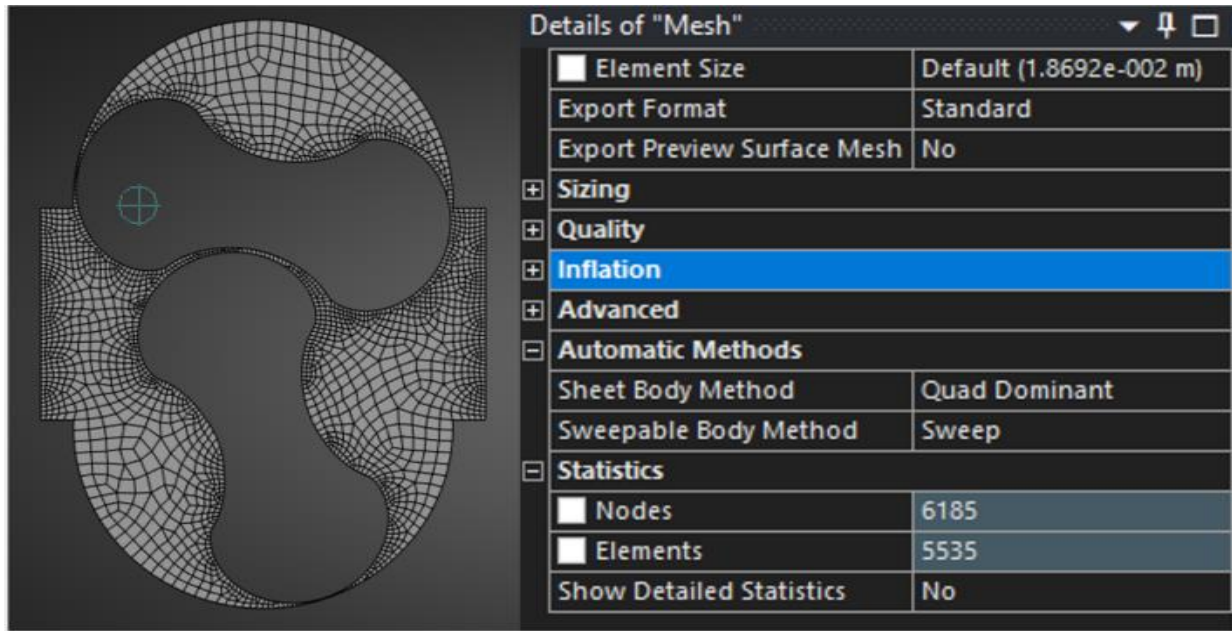


Fig. 3 Mesh details of Case-I

Meshing transforms an intricate three-dimensional object into a structured network of connected components, which creates a separate geometric model that can be used for analysis. The mesh used default element dimensions of 1.8692e-002 m to create a model that contained 6,185 nodes and 5,535 elements.

4.2. Pressure Contour Details

The pressure contour details display how pressure distributes itself throughout different parts of a geometric structure, which includes rotor blades and fluid domains during CFD analysis. Figure 4 shows the result of the pressure contour.

The updated rotor profile pressure contour demonstrates a pressure distribution that begins with high-pressure areas shown in red and extends to high-pressure areas shown in blue. The maximum pressure reaches a value of about 1.087×10^5 Pa at the rotor's compression area, which directs the working fluid toward the outlet. The rotor pressure is lowest at the intake, which is approximately 1.021×10^5 Pa.

The pressure transition in the rotor cavity shows effective compression performance as it exhibits low backflow. This shows good pressure buildup, and the rotor design is effective in maintaining sealing and pressure control during its normal operation.

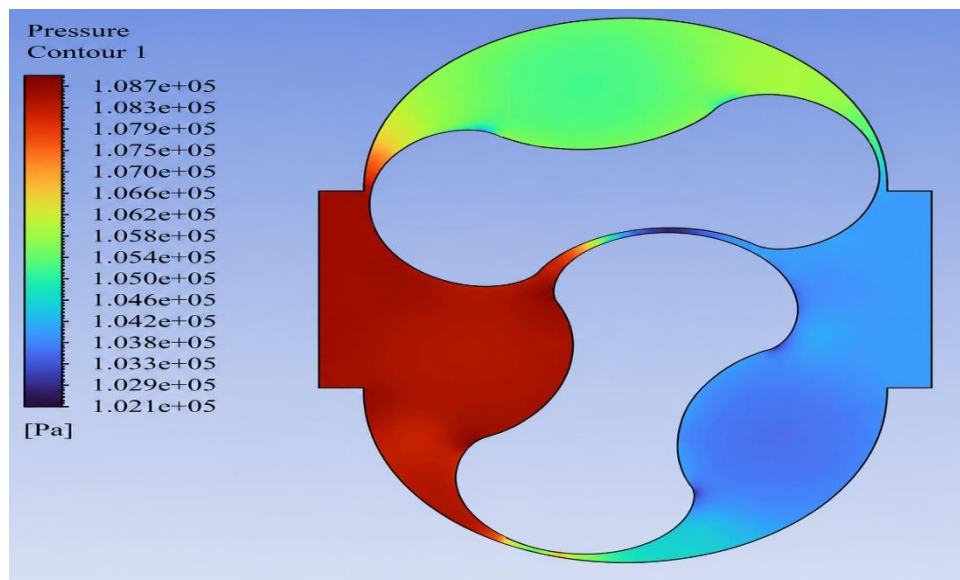


Fig. 4 Pressure contour of Case-I

4.3. Velocity Contour Details

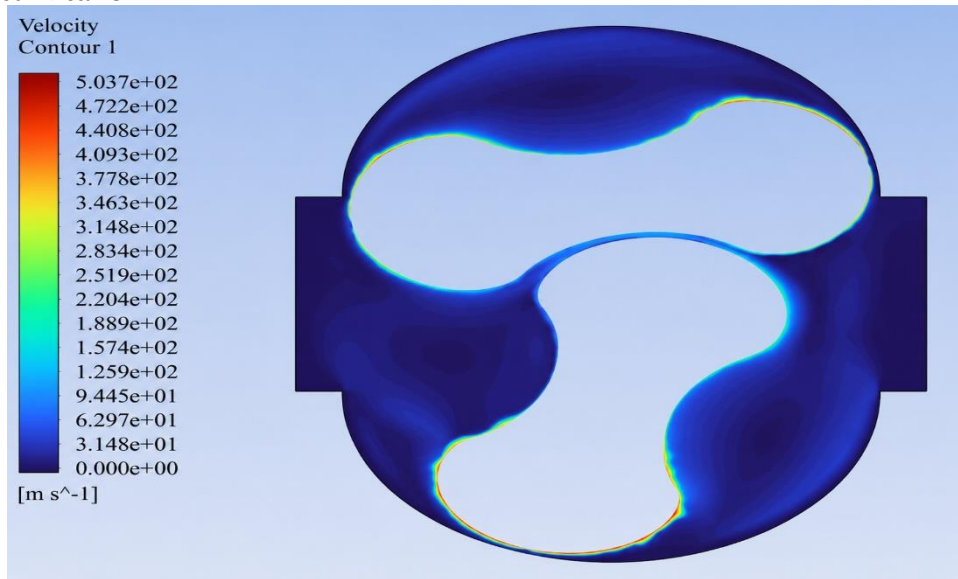


Fig. 5 Velocity contour of Case-I

The Velocity contour Details are shown in Figure 5, which displays the fluid velocity variations in the whole computational domain, including the rotor and blade passage and flow channel regions in CFD analysis. The velocity contour details are shown in Figure 5. The velocity contour displays the distribution of fluid speed throughout the main rotor design's rotor cavity. The maximum velocity reaches approximately 503.7 m/s, which occurs in narrow flow passages that exist between the rotor edges and the casing, as shown by bright red and orange regions. The high-velocity zones show locations where a narrow space exists because

flow moves at high speeds, which results from pressure-based compression. The darker regions that surround the center and wall areas show low-velocity zones that reach values close to 0 m/s, which indicate areas of stagnation or recirculation. The rotor system demonstrates effective airflow control because its low-to-high velocity gradient creates a continuous speed transition, which stops airflow from becoming turbulent. The master rotor design achieves efficient kinetic energy transfer and strong directional flow according to the contour, which enables optimized compression while achieving minimal energy losses.

4.4. Temperature Contour Details

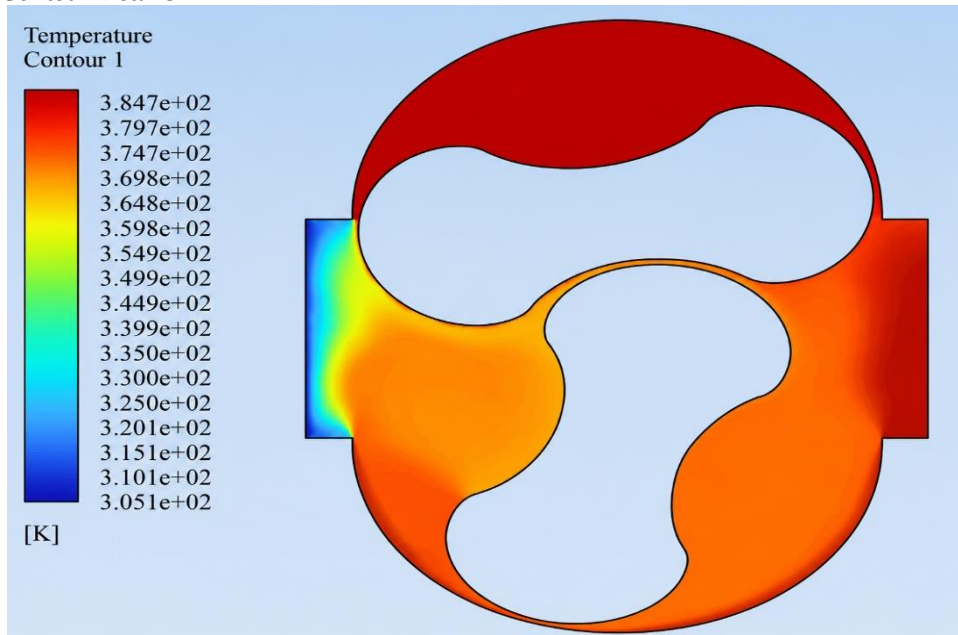


Fig. 6 Temperature contour of Case-I

The CFD and thermal analysis temperature contour details show how the temperature is distributed in the entire domain, both in the fluid region and the solid region. The temperature contour details are shown in Figure 6.

The modified temperature contour of the rotor profile illustrates the temperature distribution in the rotor cavity. The hotter areas (in dark red), with temperatures up to about 384.7 K, are located in the outlet and compression zones. The effect of the pressure generation in the compression cavity and the leakage loss during the process causes the temperature to rise. The lower temperatures, which are shown in green and blue, have cool intake regions ranging from 305.1 K.

The temperature of the fluid entering the rotor is initially lower and increases in the compression chamber. The rotor's good thermal conductivity throughout its volume is evident as it has a gradual transition from hot to cold zones. The temperature variation exhibits the thermal retention of the rotor that helps to increase the compression efficiency and decrease the energy loss.

4.5. Surface Streamlines

The surface streamlines visualize the trajectories of fluid particles flowing through different geometric surfaces that contain the rotor blades and impellers, and the duct walls during CFD simulations.

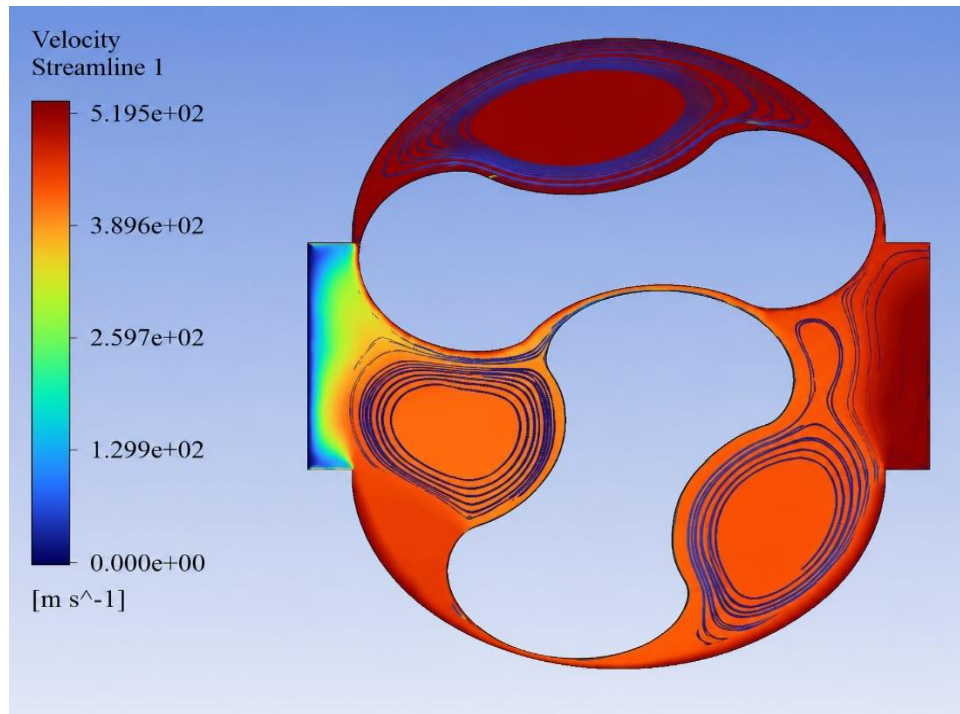


Fig. 7 Surface streamline of Case-I

The surface streamlines are presented in Figure 7. The streamline plot illustrates air flow patterns within the rotor cavity as the new rotor design allows this capability. The streamlines reveal the trajectories of the fluid particles, giving an indication of the flow field inside the cavity. The contour shows three distinct regions with high flow speeds from the inlet to the outlet, where the maximum flow speed is 519.5 m/s. The curved streamlines along the walls indicate that the flow is being efficiently guided and that there is little turbulence in these areas. The lower and left lobes form eddies, which result in recirculation zones, as the velocity falls below 0 m/s. The recirculation leads to small losses in energy, but they demonstrate the possibility of rotational motion in a confined space. The new rotor design creates a tight and uniform flow pattern that allows control over the air system, as well as maintaining stability that is crucial for efficient compression and optimum operation.

4.6. Graphical Representation

This Figure 8 shows the convergence of various solution parameters, such as continuity, x-velocity, y-velocity, energy and turbulence kinetic energy (k), specific dissipation rate (omega) for 250 iterations of the master rotor profile. The residuals are monotonically decreasing, which shows that there is a stable and accurate solution. The converged values for energy and velocity components, along with the turbulence quantities k and omega, converged to a value less than $1e-05$. The continuity residual converged at a value of $1e-03$, which is a normal value for CFD simulations of rotating machinery. The numerical simulation demonstrated stable and reliable results, and the results for streamlines and contours demonstrated physical properties. The new rotor design requires this validation approach to assess the performance of the rotor during particular operating conditions. The technical representation of this work is shown in Figure 8 below.

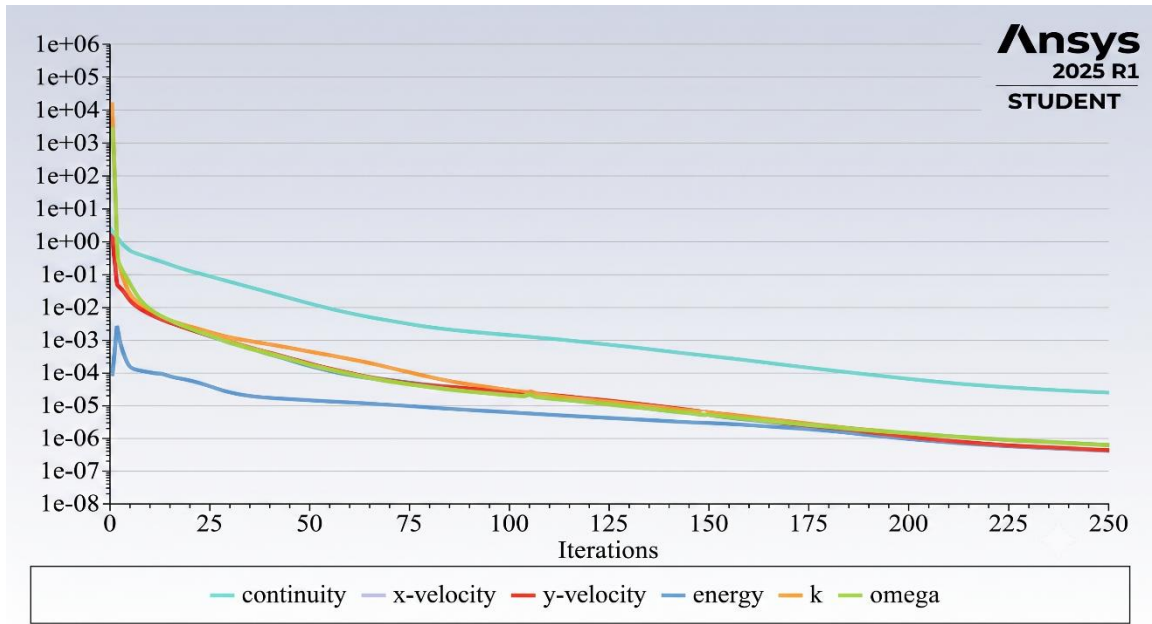


Fig. 8 Graphical representation

5. Comparative Analysis

The base circle dimensions required changes to the rotor profile design. The waist width is modified, and different models are generated accordingly with typical steps. This newly developed rotor is evaluated through numerical simulation, and its performance is compared against existing rotor designs. The results are as follows:

5.1. Pressure Contour

The modified rotors create an equal pressure distribution, which results in a steady pressure line distribution. The New CASE -I design achieves better pressure increase performance than the existing design. The system demonstrates enhanced sealing ability, which leads to improved pressure retention because it produces elevated pressure levels across the system, particularly in the areas close to compression zones.

5.2. Temperature Contour

The updated rotor temperature contour shows from its existing design as a straight line. The system maintains a

uniform temperature distribution that remains constant throughout its operation.

The system experiences elevated temperatures at its interaction zones because increased compression and thermal buildup occur when leakage rates drop.

5.3. Velocity Contour

The newly designed Case I Mambo rotors exhibited increased velocity gradients in their housing wall regions, which demonstrated improved flow control capabilities and enhanced energy transmission performance.

5.4. Graphical Representation

The new rotor case demonstrates improvement because I evaluated three performance metrics according to my design model. The graphs demonstrate that optimal efficiency results from both optimal geometric design and optimal clearance distances, which requires higher manufacturing precision and may lead to wear problems.

Table 6. Summary of the results

Blower Model	Pressure (Pa)	Velocity (m/s) (streamline contour)	Temperature (K)	Inference
Master Rotor (traditional)	1.076×10^5	514.4	384.	Baseline rotor
Modified new Rotor	1.087×10^5	519.5	384.7	Better than Master. Effective profile

Table 6 above shows the summarized results of the proposed rotor system. The modified rotor system shows improved performance according to all essential measurement standards when compared to the original master rotor system. The pressure changes from $[1.076 \times 10^5] \text{ Pa}$ to $[1.087 \times 10^5] \text{ Pa}$, which shows better compression performance. The

modified geometry results in a velocity increase from 514.4 m/s to 519.5 m/s, which demonstrates improved flow efficiency. The temperature remains almost constant because it only increased from 384 K to 384.7 K, which shows thermal stability. The modified rotor system shows better aerodynamic performance than the baseline design through its more effective aerodynamic profile.

5.5. Discussion

Computational analysis results demonstrate that the performance of a twin lobe blower is related to the rotor profile design. The new involute-based rotor profile design demonstrated better pressure retention and leakage reduction, and also better thermal performance than the conventional design. Pressure contours analysis demonstrated that system performance improved because of better distribution and sealing, which resulted in increased peak pressure in the compression zone. The velocity contours showed that increased velocity gradients occurred in the clearance areas, which allowed better flow control and energy distribution. The temperature contours demonstrated that greater heat buildup occurred due to the reduction in leakage, which enabled better compression performance but needs thermal management in practical applications. The regression analysis showed that higher degree polynomials, particularly the cubic model, are more accurate to represent the rotor profile, which allows precise geometry design required for CFD simulations. The system must be further optimized in terms of clearance volumes and profile curvature because current performance problems occur due to recirculation and small backflow. The system requires testing to validate its performance because computational simulations are used to reveal its functionality.

6. Experimental Validation

6.1. Test Architecture and Evaluation Parameters

The experimental apparatus comprised a twin-lobe Roots

blower driven by a 55 kW DC motor through a V-belt drive (SPB 4060 V-belt, center distance 117 mm, motor pulley $\varnothing 250$ mm, blower pulley $\varnothing 170$ mm). Downstream volumetric flow was quantified via a calibrated orifice plate (Orifice No. 55, pipe diameter 129.82 mm, flow threshold $< 0.4444 \text{ m}^3/\text{s}$). Primary instrumentation logged static line pressure, differential orifice head, temperature, and electrical power consumption at an operational speed of 3500 RPM. Comparative experimental validation was performed between the baseline Master Rotor and the optimized Case I Rotor (23.835 mm waist width) under identical operating boundary conditions. Measured Primary Test Data is presented in Table 7.

Table 7. Measured primary test data

Blower Model	Upstream Orifice Pressure, P3 (mbar)	Differential Pressure, H (mbar)
Master Rotor	1076	71
Case I Rotor	1084	74

7. Results and Discussion

7.1. Comparative Performance Analysis

As presented in Table 8, the experimental data confirm that optimizing the rotor profile geometry significantly enhances fluid-dynamic efficiency, pressure generation, and volumetric performance while lowering drive power demands.

Table 8. Performance metrics comparison

Evaluation Parameter	Master Rotor	Case I Rotor	Improvement
Theoretical Pressure ($\times 10^5$ Pa)	1.076	1.084	+0.74%
Simulated Pressure ($\times 10^5$ Pa)	1.076	1.087	+1.02%
Mass Flow Rate (m^3/hr)	1108	1146	+3.32%
Operating Voltage (V)	420	420	0.00%
Current Draw (A)	100	97	-3.00%
Power Consumption (kW)	42.00	40.74	-3.05%

7.2. Impact of Case I Rotor Profile Optimization

Pressure Retention: Case I achieved higher peak pressure levels (1.087×10^5 Pa simulated vs. 1.076×10^5 Pa baseline). The narrow 23.835 mm waist geometry reduced inter-lobe clearance, effectively restricting backflow leakage from the discharge port back into the suction chamber.

Volumetric Efficiency: Mass flow rate increased to $1146 \text{ m}^3/\text{hr}$ (+3.32% over Master Rotor). Enhanced internal dynamic sealing prevented fluid recirculation across the inter-chamber gaps, yielding higher air delivery.

Energy Consumption: Operating power dropped to 40.74 kW (+3.05% energy savings). Mitigating internal leakage reduced parasitic compression losses, preventing unnecessary energy expenditure to compensate for backflow volume.

7.2.1. Uncertainty Analysis

Here, statistical uncertainty evaluation was conducted for the mathematical model used to generate the rotor profile.

Geometric Model Regression Uncertainty

The 3rd-order polynomial fit used to approximate the rotor profile yielded a Coefficient of Determination (R^2) of 0.912 and a Standard Error (SE) of 1.84 mm. Following standard metrological guidelines (ISO/IEC Guide 98-3: GUM), the standard uncertainty associated with the rotor geometry prediction is:

$$u(y_{\text{profile}}) = 1.84 \text{ mm}$$

For a 95% confidence interval (coverage factor $k = 2$), the expanded geometric uncertainty is:

$$U = k \cdot u(y_{\text{profile}}) = 2 \times 1.84 = \pm 3.68 \text{ mm}$$

Based on the experimental test rig setup, the measured parameters and the complete Uncertainty Analysis for the Master Rotor and Case I are as follows-

Measurement Parameters & Instrument Precision

From the experimental test setup, the primary measured variables and their standard instrumentation/reading uncertainties (u_{inst}) are defined as follows:

Performance Metrics Comparison is presented in Table 9.

Table 9. Performance metrics comparison

Variable	Instrument / Method	Value Range (Master / Case I)	Assumed Precision (δx)	Standard Uncertainty $u(x)=3\delta x$
Upstream Pressure (P_3)	Pressure Measurement Point (P_3)	1076 – 1084 mbar	± 1.0 mbar	$u(P_3) = 0.577$ mbar
Orifice Differential (H)	Differential Pressure Measurement	71 – 74 mbar	± 0.5 mbar	$u(H) = 0.289$ mbar
Voltage (V)	Voltage Monitoring Instrument	420 V	± 1.0 V	$u(V) = 0.577$ V
Current (I)	Current Monitoring Instrument	97 – 100 A	± 0.5 A	$u(I) = 0.289$ A
Waist Width (X_o)	Precision CNC Machining / Inspection	23.835 mm (Case I)	± 0.005 mm	$u(X_o) = 0.0029$ mm

7.2.2. Uncertainty Propagation

Using the standard Kline-McClintock Method / ISO GUM Law of Propagation of Uncertainties

$$(u_c^2(y) = \sum \left(\frac{\partial y}{\partial x_i} \right)^2 u^2(x_i)):$$

Power Consumption Uncertainty Calculation ($P = V \times I$)

The power is $P = V \times I \times \sqrt{3} \cos \theta$ or $P = 420 \text{ V} \times 100 \text{ A} = 42.0 \text{ kW}$) for Master Rotor, and $420 \text{ V} \times 97 \text{ A} = 40.74 \text{ kW}$ for Case I).

Relative Uncertainty Formula:

$$\frac{u(P)}{P} = \sqrt{\left(\frac{u(V)}{V} \right)^2 + \left(\frac{u(I)}{I} \right)^2}$$

Calculation for Case I ($V = 420 \text{ V}, I = 97 \text{ A}$):

$$\begin{aligned} \frac{u(P)}{P} &= \sqrt{\left(\frac{0.577}{420} \right)^2 + \left(\frac{0.289}{97} \right)^2} = \\ &= \sqrt{(0.00137)^2 + (0.00298)^2} = \\ &= \sqrt{0.00000188 + 0.00000888} = 0.00328 \text{ (0.328\%)} \end{aligned}$$

Absolute Uncertainty for Case I Power:

$$u(P_{\text{Case I}}) = 40.74 \text{ kW} \times 0.00328 = \pm 0.134 \text{ kW}$$

Calculation for Master Rotor ($V = 420 \text{ V}, I = 100 \text{ A}$)

$$\frac{u(P)}{P} = \sqrt{\left(\frac{0.577}{420} \right)^2 + \left(\frac{0.289}{100} \right)^2} = 0.00319 \text{ (0.319\%)}$$

$$u(P_{\text{Master}}) = 42.00 \text{ kW} \times 0.00319 = \pm 0.134 \text{ kW}$$

B. Mass Flow Rate / Orifice Flow Uncertainty

$$(Q_m \propto \sqrt{H \cdot P_3})$$

According to the evaluation parameter, the mass flow rate depends directly on the square root of the differential pressure H and pressure. P_3 :

$$Q_m = K \sqrt{\left[\frac{H \times P_3}{T_3} \right]}$$

Relative Uncertainty Formula (ignoring constant T_3 drift):

$$\frac{u(Q_m)}{Q_m} = \sqrt{\left(\frac{1}{2} \frac{u(H)}{H} \right)^2 + \left(\frac{1}{2} \frac{u(P_3)}{P_3} \right)^2}$$

Calculation for Case I ($H = 74 \text{ mbar}, P_3 = 1084 \text{ mbar}, Q_m = 1146 \text{ m}^3/\text{hr}$):

$$\begin{aligned} \frac{u(Q_m)}{Q_m} &= \sqrt{\left(\frac{1}{2} \cdot \frac{0.289}{74} \right)^2 + \left(\frac{1}{2} \cdot \frac{0.577}{1084} \right)^2} = \\ &= \sqrt{(0.00195)^2 + (0.000266)^2} = 0.00197 \text{ (0.197\%)} \end{aligned}$$

Absolute Uncertainty for Case I Mass Flow Rate:

$$u(Q_{m, \text{Case I}}) = 1146 \times 0.00197 = \pm 2.26 \text{ m}^3/\text{hr}$$

Calculation for Master Rotor ($H = 71 \text{ mbar}, P_3 = 1076 \text{ mbar}, Q_m = 1108 \text{ m}^3/\text{hr}$):

$$\frac{u(Q_m)}{Q_m} = \sqrt{\left(\frac{1}{2} \cdot \frac{0.289}{71} \right)^2 + \left(\frac{1}{2} \cdot \frac{0.577}{1076} \right)^2} = 0.00205 \text{ (0.205\%)}$$

Absolute Uncertainty for Master Rotor Mass Flow Rate

$$u(Q_{m, \text{Master}}) = 1108 \times 0.00205 = \pm 2.27 \text{ m}^3/\text{hr}$$

3. Summary of Experimental Results with Expanded Uncertainty ($k = 2, 95\%$ Confidence)

Table 10. Experimental results with expanded uncertainty

Parameter	Master Rotor	Case I	Performance Improvement	Significant Within Uncertainty
Upstream Pressure (P_3)	1076 ± 1.15 mbar	1084 ± 1.15 mbar	+8 mbar (+0.74%)	Yes (8 mbar > 1.15 mbar)
Orifice Differential (H)	71 ± 0.58 mbar	74 ± 0.58 mbar	+3 mbar (+4.23%)	Yes (3 mbar > 0.58 mbar)
Mass Flow Rate (Q_m)	1108 ± 4.54 m ³ /hr	114 ± 4.52 m ³ /hr	+38 m ³ /hr (+3.32%)	Yes (38 m ³ /hr >> 4.54 m ³ /hr)
Power Consumption (P)	42.00 ± 0.27 kW	40.74 ± 0.27 kW	-1.26 kW (-3.05%)	Yes (1.26 kW >> 0.27 kW)

As presented in Table 10, Multiplying standard uncertainty by a coverage factor $k = 2$ gives the expanded uncertainty ($U = 2 \cdot u$). An uncertainty analysis using the ISO Guide to the Expression of Uncertainty in Measurement (GUM) and the Kline-McClintock error propagation method was performed on the experimental test rig data for both the Master Rotor and Case I. Mass Flow Rate Uncertainty: Based on the measurement tolerances of the orifice differential pressure ($H = \pm 0.5$ mbar) and upstream pressure ($P_3 = \pm 1.0$ mbar), the calculated expanded uncertainty ($k = 2$, 95% confidence interval) for mass flow rate is ± 4.54 m³/hr for the Master Rotor and ± 4.52 m³/hr for Case I. The measured flow improvement of Case I (1146 m³/hr) over the Master Rotor (1108 m³/hr) is +38 m³/hr (+3.32%), which significantly exceeds the experimental uncertainty boundary.

Power Consumption Uncertainty: Considering instrument accuracies for electrical voltage (± 1.0 V) and current (± 0.5 A), the expanded uncertainty in power measurement is ± 0.27 kW. The recorded power reduction for Case I (40.74 ± 0.27 kW) compared to the Master Rotor (42.00 ± 0.27 kW) demonstrates a statistically significant energy reduction of 3.05%. These calculations confirm that all reported performance gains of Case I over the Master Rotor are statistically significant and well outside experimental measurement error margins.

8. Conclusion

The primary role of twin lobe blowers in industrial systems is due to their capacity to deliver oil-free air via their reliable operation, which supports wastewater treatment, pneumatic conveying and combustion air supply systems. Three main elements contribute to the performance of a system, which are the design of a system's rotor, its air flow system, and its thermal effectiveness during operating conditions. The design approaches that are based on standard rotor profiles with empirical and reverse engineering approaches pose a barrier to the designers to develop innovative designs while trying to reach a targeted optimization goal.

The techniques used in this study are unable to accurately simulate complex flow characteristics, which comprise of leakage and recirculation and thermal effects, as they rely on basic assumptions and limited experimentation, which leads to poor performance. This study presented a new rotor design

that was created using parametric modeling and regression analysis, which defined precise coordinates for the involute-based rotor design. This research used a three-dimensional model, based on finite volume computational simulations, to determine the pressure and velocity, temperature, and streamline distributions. The experimental findings demonstrated that the newly designed rotor produces better pressure retention, low leakage, improved velocity distribution and better thermal performance in comparison to the conventional rotor design. The higher-order regression models correctly depict the rotor profile, which improves simulations and allows easy manufacturing methods by eliminating the need for reverse engineering and difficult machining processes. The optimized Case I Rotor profile demonstrates clear performance superiority over the Master Rotor baseline, simultaneously delivering higher discharge pressure, enhanced volumetric throughput, and lower electrical power draw.

Funding

On Behalf of all authors, the corresponding author states that they did not receive any funds for this project.

Conflicts of Interest

The authors declare that we have no conflict of interest.

Data Availability Statement

Primary Dataset: All primary mathematical datasets generated during this study-including the complete profile coordinate calculations across Curves 1, 2, and 3, regression model outputs, and summary CFD flow parameters-are included within this published article (Tables 1 - 6).

Code Availability Statement

Mathematical & Regression Algorithms: The coordinate point evaluations and linear/quadratic/cubic regression fittings were computed based on the explicit piecewise profile equations detailed in Section 1.

Commercial Software Configuration: Computational Fluid Dynamics (CFD) modeling was performed using ANSYS (ANSYS 2025 R1 Student Edition) via standard Finite Volume Method (FVM) solvers. All governing equations, boundary conditions, and grid parameters required to replicate the solver environment are fully detailed in Section 3.

Author's Contributions

Author 1: Performed the Analysis the overall concept, writing and editing.

Author 2: Participated in the methodology, Conceptualization, Data collection and writing the study.

References

- [1] Suraj Abdan et al., "Experimental Validation of the Screw Compressor Oil Drag Model for Various Rotor Profiles," *Proceedings of the Institution of Mechanical Engineers, Part E: Journal of Process Mechanical Engineering*, vol. 238, no. 6, pp. 2740-2749, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [2] S. Patil, "Research in Profiling and Development of a New Screw Rotor Profile," Doctoral Thesis, City, University of London, pp. 1-64, 2023. [[Google Scholar](#)] [[Publisher Link](#)]
- [3] Ildiko Brinas et al., "A Computerized Analysis of Flow Parameters for a Twin-Screw Compressor using SolidWorks Flow Simulation," *Computation*, vol. 13, no. 8, pp. 1-20, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [4] Ali Monazami, Somaye Mohammadi, and Mehdi Behzad, "Introducing a Method for Backflow Detection in Lobe Blowers through Ultrasonic Technique," *The E-Journal Of Nondestructive Testing & Ultrasonics*, vol. 29, no. 12, pp. 1-7, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [5] Robert W. Horst, "A Family of High-Flow Roots Blower Profiles based on Non-Circular Gears and the Coin Rotation Paradox," *Engineering Engrxiv Archive*, pp. 1-24, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [6] Wei Wang, Di Yan, and Chengpeng Li, "Internal Flow and Thermodynamic Performance Study of Straight and Screw Claw-Type Hydrogen Circulation Pump," *SSRN*, pp. 1-20, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [7] Brijeshkumar Patel et al., "Study of Leakage Flow in Oil-Free Positive Displacement Rotary Machines using Particle Image Velocimetry," *Experimental Thermal and Fluid Science*, vol. 145, 2023. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [8] Wenjing Lyu et al., "CFD Analysis of Single and Two-Phase Fluid Flow in a Roots Blower," *International Journal of Heat and Fluid Flow*, vol. 120, 2026. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [9] Neeraj Bikramaditya, Sham Rane, and Ahmed Kovacevic, "Shape Optimization of a Roots Blower Rotor Profile Using CFD Model with a Coupled Adjoint-Sculpting Method," *International Compressor Engineering Conference*, Purdue, pp. 1-11, 2024. [[Google Scholar](#)] [[Publisher Link](#)]
- [10] Ngoc-Tien Tran, and Duc-Minh Nguyen, "Analysis of Flow Characteristics of Cylindrical and Helical Type Multi-Lobe Roots Blower," *EUREKA: Physics and Engineering*, vol. 1, no. 1, pp. 67-75, 2023. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [11] Neeraj Bikramaditya et al., "Development of CFD Model for Radial Leakage Flow in a Roots Blower and Validation Using Particle Image Velocimetry," *Results in Engineering*, vol. 26, pp. 1-16, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [12] Leonardo Ubiola-Soto, and Adolfo Delgado, "Rotordynamic System-Level Effects of Three-Lobe Journal Bearings Including Manufacturing Variations," *Lubricants*, vol. 14, no. 2, pp. 1-19, 2026. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [13] Neeraj Bikramaditya et al., "Thermal Deformation of Radial Leakage Gaps in Roots Blower using Coupled Solvers and Dynamic Mesh," *SSRN*, 2026. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [14] Zhu Enlong et al., "Dynamic Mesh Simulation Flow Field and Micro-Filling Experiment Analysis of Viscous Medium Transported by Two-Lobe Rotary Pump," *Researchsquare*, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [15] Kai Ma et al., "Theoretical and Numerical Research on High-Speed Small Refrigeration Twin-Screw Compressor," *Applied Sciences*, vol. 15, no. 7, pp. 1-15, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [16] Filip Lisowski, and Marcin Augustyn, "Analytical and Computational Fluid Dynamics Methods for Determining the Torque and Power of a Vertical-Axis Wind Turbine with a Carousel Rotor," *Applied Sciences*, vol. 15, no. 1, pp. 1-23, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [17] Dhruilkumar Valand et al., "Investigation of the Tooth Profile for Screw Compressor Considering Leakages by Simulating in CFD," *International Journal of Advanced Research in Science, Communication and Technology*, vol. 3, no. 11, pp. 115-124, 2023. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [18] Li Zhang et al., "A Novel Design on Smooth Rotor Profile with Meshing Clearance for Screw Vacuum Pump," *Advances in Mechanical Engineering*, vol. 14, no. 1, pp. 1-11, 2022. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [19] Z.F. Huang, and Z.X. Liu, "Numerical Study of a Positive Displacement Blower," *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science*, vol. 223, no. 10, pp. 2309-2316, 2009. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [20] Antonio Giuffrida, "A Geometric Study of Different Curves for the Rotor Profiles of a Twin-Screw Compressor," *E3S Web of Conference*, vol. 312, pp. 1-11, 2021. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]