

Original Article

Performance Analysis of Various Rotor Geometries using CFD Simulation Techniques

Swati P. Shinde¹, D.B Jadhav²

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

¹Corresponding Author: shindeswati2009@gmail.com

Received: 06 March 2026

Revised: 30 April 2026

Accepted: 13 July 2026

Published: 26 October 2026

Abstract - Recently, CFD has shown it can effectively explore and fine-tune how rotors work under different conditions. Although they are dependable, the old-fashioned experimental methods can be kind of costly and oftentimes, long. Thus, this work involves a comparative CFD study to validate the thermodynamic properties and flow characteristics of various rotor profiles with a constant clearance of 0.15 mm, with a focus on flow dynamics and sealing characteristics. The study examines the variations of temperature, velocity and pressure within the rotor geometry. There are four modified profiles (Profiles 1 - 4) looked at, and the Master Rotor is essentially the reference profile or the baseline. The velocity and temperature readings under very high pressure, velocity, and temperature readings show that Profile 1 performs better than the others, thus indicating better sealing and less restrictive flow passages. Leakage (or a sort of flow separation) occurs in Profiles 2, 3 and 4, with the performance remaining in the middle to poor. Overall, Rotor Profile 1 is the best design that fits within the limits and is a good choice for applications that are more demanding, particularly if the manufacturing tolerances can consistently be maintained to keep this 0.15 mm clearance.

Keywords - Rotor, Geometry, CFD, Ansys, Simulation, Techniques, Methods, Approaches.

1. Introduction

The demand for efficiency and high performance has increased considerably in recent years, so that in recent years the planning and optimization of rotor geometries have progressed significantly [1]. Rotor is essentially a crucial part of many technical applications, such as electric motors, compressors, pumps and turbines. Rotor design really impacts its aerodynamic and hydrodynamic performance, along with flow and pressure dynamics. This influences the equipment's efficiency and energy use, too. Computational Fluid Dynamics (CFD) is a relatively robust method for examining and optimizing the performance of different rotor designs in different operating conditions [3]. There are numerous shapes of rotors, each one of them is designed for a specific task and is known as the rotating component of electrical machines. Geometries refer to the various configurations of rotors, as well as the shape of the rotors in rotating machinery, and their shapes play a significant part on the behavior and performance of the machine [4]. In electric motors, pumps, and wind turbines, you'll find typical rotor types. For example, the number and pattern of the slots in electrical appliances or the shape of blades in wind turbines have a significant impact on the torque and efficiency measures [5]. In other aspects of the performance of rotating equipment, such as torque, efficiency and stability, the shape of the rotor plays a significant role. So

it is also important to consider the overall rotor structure, bar size and the shape of the slots when considering rotor geometries [6].

FEA is a widely used method for evaluating these effects and allows for rotor design improvements for specific applications. Fluid rotors, strong poles, squirrel cages, winding, and permanent magnets are the primary varieties. Every design offers distinct benefits in terms of economy, speed, and torque [7].

1.1. Types of Rotors in Rotating Equipment

1.1.1. Squirrel Cage Rotor

This is the kind that is mostly employed by induction motors. Laminated cores are made up of bars, which are usually made of copper and sometimes aluminum. The whole arrangement is somehow in the form of a squirrel cage, hence its name. It is also referred to as one that is easy to employ, sturdy and has a high starting torque [8].

1.1.2. Wound Rotor

The same can be said about the induction motors as well, but there, the rotor is different since it uses a stator type of winding, which is normally a Y connection. Here, the rotor shaft has slip rings which are connected to the winding and



external resistances can be added to control the torque/speed characteristic curve. It gives more control of the motor behavior [9].

1.1.3. Salient Pole Rotor

Utilized most often in synchronous machines, especially those that operate at slower rates. distinguished by conspicuous poles that protrude from the rotor surface. Applications requiring high torque at low speeds may use this design [10].

1.1.4. Permanent Magnet Rotor

Consists of rotor-mounted permanent magnets that generate the magnetic field. frequently utilized in applications that operate at high because to its great power density & effectiveness. Performance depends on the magnets' design and material selection [11].

1.1.5. Fluid Rotor

Uses a fluid medium within the rotor for a number of functions, including actuation and cooling. The fluid and how it interacts with the rotor shape are determined by specific applications.

1.1.6. Cylindrical Rotor

Non-salient pole rotors are another name for them. has integrated windings and a smooth, cylindrical surface. usually seen in synchronous machinery at high speeds, such as turbo generators [12].

1.1.7. Built-Up Rotor

Comprises of a shaft with discs that have been shrunk and keyed onto it; it is used in steam turbines. Ideal for uses where a high-power output is required.

1.1.8. Integral Rotor

Wheels & shaft are one forging. An additional steam turbine design that provides great strength and dependability.

1.1.9. Hollow Drum Rotor

Helps maintain thermal stability by promoting a uniform temperature distribution. used in steam turbines, particularly those at high speeds.

1.1.10. Solid Drum Rotor

There will be differences in torque, efficiency, durability and other characteristics, but not as significantly as one might expect where the range of temperatures is not large [13]. A V+1 rotor design, for instance, may give a reduction in no-load harmonics, cogging torque, increased torque production, and increased efficiency over other rotor configurations used in the switching reluctance motors. Power factor and operating range for wind turbines would greatly depend on rotor blade pitch angle and the number of blades. In addition, rotor bar design and air gap size are factors affecting torque ripple, power factor and efficiency of induction motors [14].

1.2. Different Rotor Geometries Impact Performance

1.2.1. Switched Reluctance Motors

V+1 rotor form has been shown to have superior efficiency due to high temperatures, decreased cogging torque, increased output torque, and reduced harmonics at zero loading conditions. However, despite all these, there are other rotor geometries which may be preferred for other reasons. This is because what makes the “perfect” rotor geometry would totally depend on the desired application and the desired properties [15].

1.2.2. Wind Turbines

Higher peak power coefficients (C_{pMAX}) may result from a bigger pitch angle (e.g., $\beta = 65^\circ$), but the operating range may be less. According to the rotor design and operating circumstances, three-bladed VAWTs may be more advantageous than two-bladed ones in certain ways, while four-bladed rotors may be less efficient than two-bladed ones. These factors have a major effect on the blades' performance and aerodynamics, which in turn impact lift, drag forces and, eventually, power production [16].

1.2.3. Induction Motors

Although they may result in greater production costs and a larger danger of rotor-stator contact, smaller air gaps often enhance power factor and lower no-load losses. The motor's torque ripple, effectiveness, and other indicators of performance are influenced by the rotor bars' configuration and shape. The motor's effectiveness, power factor, & torque may all be impacted by the rotor cage's material [17].

1.2.4. Other Rotor Types

Salient pole rotors are appropriate at lower speeds because of their projecting poles, greater diameters, & shorter axial lengths. Cylindrical rotors have a greater axial length & less diameter because of their drum-shaped, parallel winding grooves. Built-up, integrated, hollow drum & solid drum rotors are common varieties; each has unique benefits for various uses and operating environments [18]. The desired performance properties, intended use, and limitations, such as cost and manufacturability, can all heavily influence the best rotor configuration. When looking to maximize the performance of any rotating machine, these considerations must be assessed thoroughly [19].

A rotor's geometry has a great influence on performance in any moniker of an application, whether it be motors, turbines, or other rotating systems. A variety of rotor configurations will greatly impact the amount of torque, efficiency, cogging torque, start torque, and ultimate power output. Analysis for assessment usually includes numerical methods such as simulation (e.g., Simulink) and Finite Element Analysis (FEA) [20]. Rotor performance as a whole can be defined as the efficiency in which a rotor can convert energy in rotational motion, or extract energy from a rotating system. Generally, is quantified by torque, power, efficiency,

& speed. Usually, values are reported for certain applications such as electric motors or wind turbines. For example, rotor efficiency in a wind turbine is quantified by the amount of power produced (output) at a certain speed from the wind (input), whereas in an electric motor, it is quantified by how much speed & torque (output) are produced at a certain electrical input [21].

There are many factors to consider when assessing the performance of a rotor. The overlap ratio of the blades is also a consideration; for Savonius rotors, for example, a larger overlap may provide more static torque. Likewise, there are many factors, namely the interaction of the stator windings and the currents produced in the rotor bars, that greatly influence the torque production of electric motors. Again, rotor dynamics include stability, vibration, and the influences of damping and internal friction, in particular for flexible rotors [22].

Rotor performance can be evaluated using a variety of methods. Some are analytical methods, such as the Blade Element Momentum (BEM) theory, which is a method for the aerodynamic design of rotor blades. Electric motor rotors can be modelled to examine rotor behaviour and simulate the electromagnetic fields using Finite Element Analysis (FEM). Experimental testing is equally important; measuring torque, speed, current, and other characteristics of interest for various operating conditions [23].

Apart from that, various alternative designs of the rotor have been considered and adjusted for enhanced performance, such as in wind turbines and marine propellers, using Computational Fluid Dynamics (CFD) models. The coefficients of the rotor power, its torque and efficiency can be obtained by CFD modeling of the fluid flow around the rotor, and CFD can assist engineers in designing more efficient and powerful rotors suited for the particular task [1].

CFD can provide very detailed flow information, including velocity, pressure and turbulence that is crucial for the analysis of rotor interaction with the flow. It may seem like an obvious step, but it is rather important to be aware of the flow pattern behavior, where separation starts, where recirculation occurs, and how performance will be affected in case there is some behavior that was not anticipated. Also, CFD can be employed in flow behavior modeling and lift, drag and other forces prediction [2]. Since there are no costly and time-intensive physical prototype tests that need to be conducted, the engineer can use CFD simulations to test different rotor designs and shapes. The engineer can use parametric studies through CFD, for example, testing the effect of parameters on efficiency, like the pitch angle or the blade shape. The engineer can then decide which rotor configuration is the most efficient and the best suited for the particular study by analyzing the CFD simulations of different configurations.

Through CFD modelling, the engineer will also be able to visualize how the rotor performs in a range of operating conditions, such as variation in rotor speed, angle of attack and even different operating environments, thus giving more practical insight into rotor performance and enabling optimization of rotor efficiency over a wide range of operating conditions. Some of the flow characteristics that are hard to achieve through conventional approaches, such as stalling characteristics, tip vortex and unsteady flows, can be easily captured using CFD approach [4]. Once these effects are known to engineers, they can come up with rotors that stall less, require less power and operate more efficiently. Thus, CFD becomes a very important instrument in terms of optimizing the performance and prediction of rotor operations. Moreover, there is an increase in the number of CFD methods being applied in rotor studies. The reasons behind such trend include high precision, lower costs and visualization of complex flow patterns. Now, there is a more systematic approach to CFD analysis of different types of rotor configurations, including straight, curved, helical and biomimetic rotor configurations, taking into account numerous performance parameters, such as pressure drop, velocity profile, turbulence and efficiency. There are many companies and researchers who rely on CFD modeling to evaluate the performance of their rotors prior to constructing prototypes [5].

In addition to this, advancements in computing capabilities and turbulence modeling, along with meshing methods, have contributed towards achieving accuracy and details in rotor analyses. Several studies in recent times focus on comparing different physical models with the objective to narrow down the rotor designs that can provide optimum performance for some particular applications, like mixing fluids, speed efficiency, and reduction in noise levels. The overall trend of using CFD in rotor analysis shows that CFD can be a powerful tool for developing and designing rotors [6].

Despite the many developments in the CFD-based analysis of rotating machines, there have been very few studies on the aerodynamic performance of these machines at ultra-tight micro-clearances. The existing literature has only concentrated on performance characteristics of devices based on standard clearance and steady-state analysis. There is no much research done on ultra-tight clearances due to which there is an inadequate understanding of how small geometric variations affect the leakage flow, pressure differential and wall shear behavior. Moreover, there is no research conducted on different rotors performing under the same micro-clearance.

Due to the lack of information on this topic, it is difficult to optimize the geometry of rotors that are capable of maintaining low leakage while still being able to create efficient pressure and maintain uniform flow.

The novelty in the research arises from the multi-parameter geometric optimization of the twin-lobe/positive displacement rotor device at highly constrained micro-clearance constraints (0.15 mm). Novelty in the research arises from:

- Optimizing Micro-Clearance Limits: Understanding how the clearances in the system affect its thermodynamics and fluid dynamics during the change from 0.5 mm clearance to a precise 0.15 mm boundary layer.
- Modifying Locally Waist Width Dimensions: Separately isolating and changing the locally waist width dimensions (from 23.835 mm to 23.910 mm in Cases I to IV) to develop an optimized geometry that performs better than conventional master rotors.
- Simultaneously Sealing and Retaining Thermal Energy: Shifting from general aerodynamic lift and drag studies to the study of minimizing back flow and leakage using local temperature differences and local wall velocities.

2. Literature Review

Scientists have been able to model the more complicated flow situations more accurately with the use of high-performance computing and the creation of other CFD tools. Several studies have examined the effects of changes to rotor blade geometry, including pitch angle, blade curvature, twist, and tip design on performance parameters such as lift, drag, torque, and efficiency. These studies provide ways to improve overall system reliability, reduce noise, and improve aerodynamic efficiency. The aim of the literature is to evaluate the current body of information about the performance analysis of different rotor geometries using CFD. It emphasizes the approaches used, important discoveries, knowledge gaps, and new developments that will direct rotor planning and optimization research in the future.

N.E.B. Afane, et.al. (2023), the Jeffcott rotor shaft system, which has dimensions of 32 mm in diameter, 1000 mm in length. This was done for a case that is simply supported. Determining the natural frequencies, critical speeds, and amplitudes (von Mises stress, primary stress, maximum shear stress, lifespan) of the rotor system is essential. According to the results of the research, structural steels and Ti-6Al-4V were responsible for the higher critical speeds for 38CrMoAl steel (1858.7, 1859.4, and 8468.1 rpm) and the lower critical speeds for JIS-S45C steels (1784.2, 1784.9, and 8147.7 rpm). For rotary machines that are exposed to high levels of stress and distortion, 38CrMoAl steels are recommended. For rotary equipment that is exposed to higher operating speeds, such as compressors, turbines, and propeller shafts, JIS S45C and 38CrMoAl are recommended.

Drusilla Duty, et.al. (2023) analyze the VAWT's flow characteristics and performance utilizing CFD simulation. The

assessment simulates turbine shape, boundary conditions, and mesh & solver settings to provide insight into the pressure and velocity spread around the turbine. The conclusion was that the current simulation model may provide valuable insight to aid in the design and development of the turbine and accurately predict the performance of a Vertical-Axis Wind Turbine (VAWT). Furthermore, it was shown that the model's coefficient of conversion increased as the TSR increased, peaking at around 2 TSR before declining. However, the torque coefficient decreases as the TSR increases. Overall, the study demonstrates the value of CFD in assessing and developing VAWT for wind power applications.

Djamal Hissein Didane, et.al. (2022) evaluate the performance of a Savonius rotor wind turbine. There are differences Savonius wind turbine blade designs, and their effects are mentioned. In total, three distinct turbine blades are modeled, with variations in the rotor's height and diameter. To conduct the simulation examination, FLUENT 2021 R21 is used. 9.2 m/s has been selected as the constant wind speed for the experiment. For the simulation, a transient temporal flow with a steady upstream wind speed was used. As per the results, all models' power coefficients increase with TSR, and the optimal efficacy is generally reached at around unity (0.9) TSR. Model 2 generates the maximum power coefficient when comparing the performances of all models, followed by Model 3 and Model 1, in that order.

Shlok Anand, et.al. (2020) assessed a computational study of two-dimensional rotor blades using ANSYS. An important part of every rocket engine is the turbo-pump. The airfoil portion of a rotor blade of the Space Shuttle Main Engine (SSME) was modeled along with a far-field region of the appropriate shape and length. The roughly appropriate discretization and refinement of the model were completed. To see if the rotor blade's performance has improved. The best inlet velocity for the optimum operating of the blade was calculated.

Arian Hosseini, et.al. (2019) develop, simulate, and evaluate the performance of a leading-edge hybrid VAWT for the purposes of increasing the operational range and self-starting ability of the turbine. A great deal of research was conducted to determine the optimum design parameters of a hybrid VAWT in its special configuration. A CFD analysis assessed the rotor system hybrid VAWT by breaking down the geometry to create the grid in ANSYS Meshing for conducting calculations using CFD (Computational Fluid Dynamics).

According to the results, the Darrieus turbine had the highest Coefficient of Power (C_p) at a TSR of 2.50, 48.4%; however, the Darrieus turbine needed considerable torque to start up. The recommended configuration demonstrated some performance improvements and operating range over previous hybrid VAWT systems, suggesting these can lead to energy savings.

Wenbo Shi, et. al. (2018), the juxtaposition of the SRP and CROR flow fields allows a comprehensive analysis of the blades aerodynamic interaction. The results gained from the URANS method also lead to an understanding of the disordered flow field produced by a CROR configuration. It is the mutual interactions of blade tip vortices that drive these aerodynamic interactions. When aerodynamic interactions occur, there are periodic oscillations on the aerodynamic forces produced, and the rate of these oscillations is connected to the number of blades. This URANS method, in conjunction with a dynamic patched grid method, is a valuable and powerful tool for analyzing propeller aerodynamic interactions. Nik Ahmad Ridhwan Nik Mohd, et.al. (2017) examined the steady Navier-Stokes Averaged by Reynolds Solver for Computational Fluid Dynamics Rotor achievement in axial flight was predicted using Helicopter Multi-Block. This study's primary goals were to verify the flow solver and further the fundamental understanding of the topic. The results are shown as vortex wake trajectories, surface pressure, and rotor performance metrics. When compared to the relevant experimental data, the projected findings for hover at descent rate exhibited a reasonable degree of consistency, indicating an unstable solution for rotors in the vortex-ring condition.

M Anbarsooz (2016) the aerodynamic efficiency of helical Savonius rotors has been investigated experimentally and computationally, and contrasted it with that of a normal rotor with zero twist. The ANSYS Using the sliding-mesh method, computer simulations that account for the turbine's rotation have been carried out using Fluent commercial software. The results show that the greatest power coefficient of helical rotors is lower than that of the conventional rotor; the coefficient decreases. But with helical rotors, the torque coefficient changes more evenly over time. At a tip speed ratio of 0.7, all rotors have achieved their maximum power coefficient. The great agreement between the computational and observational data shows the value of the aerodynamic efficiency of Savonius-type turbines.

G.N. Barakos, et.al. (2016), the anhedral tip was found to provide the highest Figure of Merit. The findings showed that aeroelastic impacts were more noticeable in situations with high thrust. Last but not least, an acoustic investigation of both rotors' tip-path-plane revealed outstanding agreement with the hypothesis about total loading noise thickness. For the anhedral tip of the model-scale blade, a 5% reduction in noise level was expected. Overall, the high degree of agreement between theory and evidence demonstrated that the existing CFD method is capable of accurately predicting rotor flows. M.H. Mohamed, et.al. (2015), in order to enhance this wind turbine's overall performance, a thorough numerical evaluation is presented in this paper. A final numerical sequence for the Darrieus rotor efficiency is summarized by comparing the meshing tools for computational modeling in ANSYS Workbench and Gambit. To get the optimal performance, this design sequence is then applied to various

blade airfoils. Unsteady simulations using URANS turbulence calculations utilizing a sliding mesh technique were carried out for various speed ratios. The SST Komega model improves ANSYS Workbench meshing accuracy, according to the results, although it is not advised for use with other turbulence models. Additionally, this article uses this CFD approach to evaluate the turbine performance using 25 various airfoil forms. With a 10% increase in efficiency above standard airfoils, the findings provided novel forms for this turbine. Furthermore, research on blade pitch angle has shown that the zero-pitch angle provides the highest efficiency.

Pinku Debnath, et.al. (2014) Power & torque coefficients at various tip speed ratios were predicted using a CFD study. Additionally, the performance of bladed helical Savonius rotors was examined at rotor angles as well as at tip speed ratios. To provide optimal meshing accuracy and computational outcomes, a 3D unstructured grid was created. The velocity and pressure contour analysis was performed using a conventional wall function & a RNG k- ϵ turbulence model. Petros Chasapogiannis, et.al. (2014), a 1.05D center-to-center spacing is used to represent the seven-rotor system under investigation. Under nominal conditions (tip speed ratio=9), predict a ~3% increase in power and a ~1.5% increase in thrust. According to the flow field analysis, the seven-rotor system's many wakes merge into a single wake at around 2D, and flow recovery starts at about the same downstream distance as when there is just one rotor. It was found that, under rated conditions, the loading amplitude rose by 0.30-2.13% at blade level due to the dynamic impacts of the rotors' close spacing.

In the analysis of, the research gap can be highlighted based on three points, which include:

- No comparative micro-clearance studies: Although Computational Fluid Dynamics (CFD) is extensively applied for analyzing and optimizing typical rotor shapes, the present literature does not include sufficient and comparative studies regarding the effect of extremely low clearance values, like a constant 0.15 mm, on simultaneous fluid dynamics and thermodynamics in different geometrical shapes.
- Focusing at Wrong Scale: Almost all the existing studies focus on higher macro-clearance scale levels or observe only one profile. This greatly reduces the academic knowledge of how tiny geometric variations affect flow characteristics, including the transition of velocity, pressure, temperature, and fluid sealing performance.
- Misalignment with Actual Production: The current CFD analysis does not cover sealing performance and flow homogeneity in high-performing configurations where efficiency depends on micro-variations. Moreover, there is a significant literature gap regarding the validation of CFD results with actual production capabilities, including whether a tight clearance of 0.15 mm can be maintained in practical operations.

Table 1. Literature comparison matrix

Author	Contributions	Limitations
N.E.B. Afane et al. (2023)	Evaluated rotor material performance comprehensively	Limited Simply Supported Rotor configuration
Drusilla Duty et al. (2023)	Assessed VAWT Performance using CFD	Experimental validation not conducted adequately
Djamal Hissein Didane et al. (2022)	Compared Savonius blade design performance	Single wind Speed Considered Only
Shlok Anand et al. (2020)	Optimized rotor blade inlet velocity	Two Dimensional analysis limits accuracy
Arian Hosseini et al. (2019)	Developed Hybrid VAWT Improving startup	Prototype validation not experimentally performed
Wenbo Shi et al.(2018)	Analyzed Propeller Aerodynamic Interaction mechanisms	Computationally intensive URANS Simulation Approach
Nik Ahmad Ridhwan Nik Mohd et al. (2017)	Predicted axial rotor aerodynamic performance	Vortex-ring Instability Remains Unresolved
M.Anbarsooz (2016)	Evaluated Helical Savonius Aerodynamic efficiency	Lower Maximum Power Coefficient Observed
G.N. Barakos et al. (2016)	Predicted Rotor Aeroacoustic Performance accurately	High-thrust Conditions Only investigated
M.H. Mohamed et al. (2015)	Optimized Darrieus airfoil CFD performance	Other Turbulence Models Unsuitable Tested
Pinku Debnath et al. (2014)	Predicted Savonius Performance using CFD	Experiment Verification not clearly presented
Petros et al. (2014)	Investigated multi-rotor wake interactions	Close Spacing Effects Require Further Validation

2.1. Research Gap

Although CFD plays an important role in the study of the fluid dynamics and thermodynamics of a number of rotor types-straight, helical, and curved, for example-the absence of comparative analysis that would show the difference in the performance of rotors with modified geometries operating at ultra-tight micro-clearances is one of the main shortcomings in current literature.

Indeed, the overwhelming majority of studies in this area concentrate on either a single type of geometry and/or larger clearances, which renders changes at the microscopic level unimportant from the viewpoint of performance. This results in the insufficient comprehension of the influence of even slight modifications in geometry on sealing properties, transition velocities, and thermal performance at ultra-tight clearances, say, 0.15 mm. Besides, there is an evident inconsistency between CFD-based optimizations and manufacturing limitations concerning ultra-tight clearances.

2.2. Problem Statement

One of the difficulties faced during the development of efficient positive displacement and rotating mechanisms lies in controlling fluid leakage and heat generation within them. The efficiency of these machines, such as torque and pressure generation, largely depends on the clearance between the rotating and fixed housing. The conventional techniques employed for assessing variations in rotor geometry are

accurate but quite costly and labor-intensive. Though numerical modeling makes it possible to quickly test several rotor geometries, the simulation of fluid dynamics transition, velocity gradients, and thermal variation within micro clearance becomes difficult to achieve. An incorrect optimization of the geometric shape of the rotor may lead to fluid stagnation, local circulation, flow separation, and even backflow, which directly influences the compression performance.

2.3. Objectives of the Present Study

In order to overcome these drawbacks, this paper introduces a detailed, comparative CFD study that will analyze the fluid dynamic and thermodynamic properties of different rotor modifications. With similar boundary conditions and with a fixed micro-clearance value equal to 0.15 mm, four different rotor modifications (Profile 1 to Profile 4) will be compared to the base "Master Rotor" configuration.

With the help of analyzing exact temperature distributions, pressure fields, fluid velocity and surface streamlines, this study is intended to demonstrate how different micro-geometric modifications affect the flow uniformity and sealing performance. In conclusion, it is necessary to justify the optimal rotor design that ensures maximum efficiency of energy exchange and fluid entrapment.

3. Methodology

This technique aims to systematically study the impact of changes in the design of rotor shape design along with design parameters. CFD can be used as a practical and cost-effective substitute for physical testing through a virtual space that simulates behavior of fluid flows around complex geometry. This research is about designing different shapes of rotors using CAD software and doing simulation tests with comparable boundary and flow conditions. The selection between these two techniques will depend on the way the time-dependent physical phenomenon, such as rotor rotation, pulsations and turbulence eddies, are taken into account by the

solver. Following is a clear explanation of the two approaches and their respective parameter tables.

- **Steady-State Simulations:** Steady state simulation means that at each point in the spatial domain, the fluid properties do not change with respect to time. The solver finds the final, fully developed equilibrium state in which all mass, momentum and energy fluxes cancel each other.
- **Transient (Time Dependent) Simulations:** Transient simulation explicitly solves the Navier-Stokes equation as a function of time. The fluid domain changes at distinct chronological intervals. Thus, you are able to observe the transition from an initial state to a continuous state.

Table 2. Key parameter summary

Governing Parameter/Feature	Steady-State Simulation	Transient (Time-Dependent) Simulation	Relevance to Current Study
Time Derivative	Set to zero	Explicitly calculated	Determines whether time variations are tracked.
Rotor Modelling Technique	Moving Reference Frame (MRF) / Frozen Rotor	Sliding Mesh Model (SMM) / Dynamic Mesh	Defines whether the rotor mesh stands still or physically rotates.
Time-Step Size	Not applicable (Iteration-based convergence)	Crucial; calculated via Courant number	Dictates temporal resolution and stability in transient runs.
Flow Physics Captured	Time-averaged flow paths, overall pressure delta, stable temperature fields.	Rotor-stator interactions, vortex shedding, pressure pulsations, transient backflow jets.	Steady-state isolates the profile shape effect; transient captures operational vibration/pulsation.
Boundary Conditions	Constant values (e.g., fixed inlet/outlet pressures)	Can be time-varying or pulsating (e.g., standard piston outputs)	Defines how the system interacts with external industrial networks.
Computational Resource Cost	Low to Moderate (Rapid turnaround for screening geometries)	Exceptionally High (Requires significant RAM, storage, and CPU/GPU hours)	Explains why steady-state is chosen for initial multi-profile comparisons (Master vs. Profiles 1 - 4).
Convergence Criteria	Residuals drop below a flat target threshold	Residuals must converge within each individual time step before advancing.	Ensures numerical accuracy and prevents solution divergence

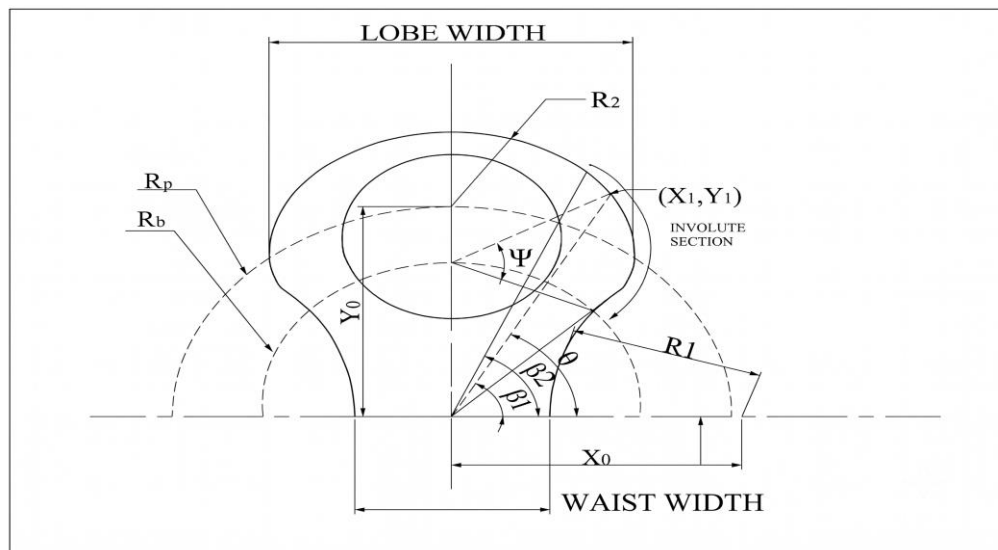


Fig. 1 Typical sketch of rotor profile

In this section, the method used to construct each curve is explained.

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

This corresponding part of the rotor profile is created by using the following two equations:

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

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

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 $\theta = [\beta_2, \pi/2]$

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

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

The design of the rotor form is done using the method of parametric design, where different equations are used for different angular ranges on the form of the rotor. When all these curves are combined, we get the complete rotor form as depicted in Figure 1.

The parameters like beta (β), including β_1 and β_2 in Table 1, play an important role in providing transition angles for the end of the curves.

Table 3. Coordinate calculations for varying θ

θ (degrees)	R_1	Cos $\{\pi - (5\theta)/4\}$	Sin $\{\pi - (5\theta)/4\}$	x_0	x_1 ($R_1 \times \cos$)	x_2 ($x_1 + x_0$)	y_1 ($R_1 \times \sin$)
0	35.2400	-1.0000	-0.0001	59.075	23.8350	23.8350	-0.0034
0.25	35.2400	-1.0000	0.0054	59.075	23.8355	23.8355	0.1887
0.5	35.2400	-0.9999	0.0108	59.075	23.8371	23.8371	0.3809
0.75	35.2400	-0.9999	0.0163	59.075	23.8397	23.8397	0.5731
1.0	35.2400	-0.9998	0.0217	59.075	23.8433	23.8433	0.7652
1.25	35.2400	-0.9996	0.0272	59.075	23.8480	23.8480	0.9573
1.5	35.2400	-0.9995	0.0326	59.075	23.8537	23.8537	1.1494
1.75	35.2400	-0.9993	0.0381	59.075	23.8605	23.8605	1.3414
2.0	35.2400	-0.9991	0.0435	59.075	23.8684	23.8684	1.5334

θ (Theta): Input angle in degrees converted in radian in the calculations for calculating trigonometric function values wherever needed, ranging from 0 to 2.0 in increments of 0.25. R_1 : Waist circle radius ($R = 35.24$ mm), assumed constant for all θ . $\cos \{\pi - (5\theta)/4\}$ & $\sin \{\pi - (5\theta)/4\}$: Used to determine position coordinates based on the angle. x_0 : Fixed value = 59.075 mm. $x_1 = R_1 \times \cos \{\pi - (5\theta)/4\}$: x-component of radius projection. $x_2 = x_1 + x_0$: Final x-coordinate position (sum of base and projected component). $y_1 = R_1 \times \sin \{\pi - (5\theta)/4\}$: y-component of radius projection, determines vertical offset.

For different angular parameter values θ , ranging between 0 to 2 radians with a step size of 0.25 radians, the coordinates have been computed and presented in tabular form. For calculating the coordinates related to the waist circle of radius ($R_1 = 35.24$ mm), the expressions $\cos \{\pi - (5\theta)/4\}$ and $\sin \{\pi - (5\theta)/4\}$ have been evaluated for every angular parameter value of θ . The values of cosine have

been multiplied by R_1 , and then an addition of $x_0 = 59.075$ mm was performed to calculate the value of x_2 coordinates.

Likewise, R_1 was multiplied by the sine of the angle equation to get the y-component (y_1). The findings indicate that the cosine value gradually increases (becomes less negative) as θ increases, leading to a progressive increase in x_1 and x_2 . This suggests a gradual shift to the right along the x-axis. The y_1 values grow steadily while the sine value also rises from a little negative value to a positive one at the same time. A steady rising movement along the y-axis is seen in this trend. Therefore, when θ rises, the calculated coordinates (x_2 , y_1) follow a smooth upward-right trajectory. In kinematic or gear profile modelling situations, the little incremental changes in the x and y components show a consistent rotational transformation according to the specified angle function.

In that case, it could be stated that the basis of the technology under study could be considered finite volume

analysis in terms of computer simulation. There were two clearances between the rotor and the casing (0.15 mm and 0.5 mm). The “Master Rotor” concept was used with the help of the following: The simulations were performed with constant inlet or shipping pressure (nearly 1.035×10^5 Pa), and with a constant speed of 3500 RPM, so that there would

be an accurate geometry mapping. They analyzed the temperature variation, velocity vectors, and the air flow performance for the respective runs through the profiles under consideration. The plots of pressure, temperature, and velocity were made for each clearance case and were only compared qualitatively.

Table 4. Properties used for master rotor profile

Master Rotor Profile		
Parameters	Input	Output
Speed	3500 RPM	3500 RPM
Delivery Pressure	1.013×10^5 Pa (1013 m bar)	1.035×10^5 Pa (1035 m bar)
Air Flow	0.00	0.3750 m ³ /s (1350 m ³ /hr)
Temp. Difference	32°C	114°C
Air density	1.2924	1.2924

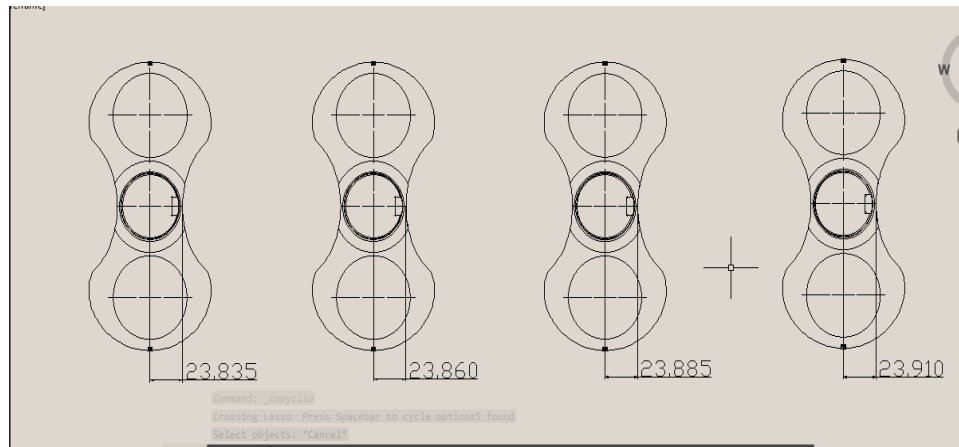


Fig. 2 Modified rotor profile dimension details

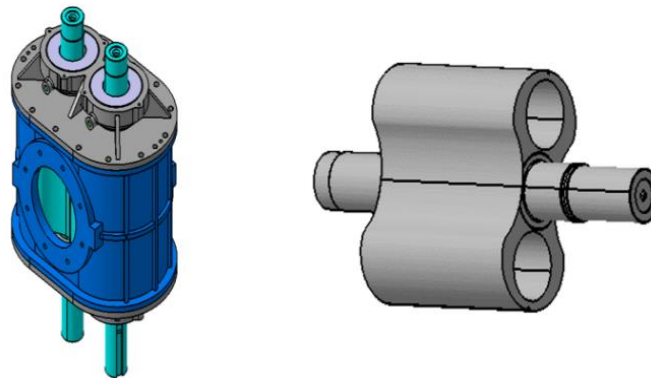


Fig. 3 Master rotor assembly profile

Table 5. Geometry variations for four different cases

Case	Waist Width Dimensions (mm)
Rotor profile 1 (Case I)	23.835
Rotor profile 2 (Case II)	23.860
Rotor profile 3 (Case III)	23.885
Rotor profile 4 (Case IV)	23.910

4. Analysis and Results

- The simulation was conducted using ANSYS software, utilizing a 3D unstructured grid for the mesh.
- Turbulence Model: The study employed the k- ω turbulence model alongside a conventional wall function for analyzing velocity and pressure contours.
- Mesh Configuration: The model was created with a default element size of approximately 1.8692e-002 m, resulting in a mesh consisting of 5,535 elements and 6,185 nodes.
- Convergence Criteria: Results were monitored through residuals, where energy, k, and omega residuals demonstrated continuous convergence below 10^{-3} , while momentum equations (x- and y-velocity) stabilized below 10^{-4} .
- Refinement Near Clearance Regions
- Localized Gradient Analysis: While the general mesh is unstructured, the manuscript highlights that "tighter color changes" in the results near curved edges and the 0.15 mm small gaps hint at complex fluid interactions that were captured by the grid.

- Sealing Effectiveness: The refinement in these critical regions allowed for the detection of sharp velocity and pressure gradients, which were used to differentiate between the sealing quality of the four altered profiles.

Utilizing ANSYS software, the study was conducted with an emphasis on assessing temperature distributions, pressure contours, velocity contours, and surface streamlines for both the revised rotor profiles and the master rotor profile. The most effective arrangement was identified by comparing the outcomes of all models. The typical sketch of the rotor profile -1 is shown in Figure 4.

4.1. Analysis and Results for Master Rotor Profile

The sketch of the Mesh details is shown in Figure 4. A complicated 3D model is meshed into a network of linked parts.

With the default element size of 1.8692e-002 m, the mesh was created, producing a model with 5,535 elements and 6,185 nodes.

Mesh details:

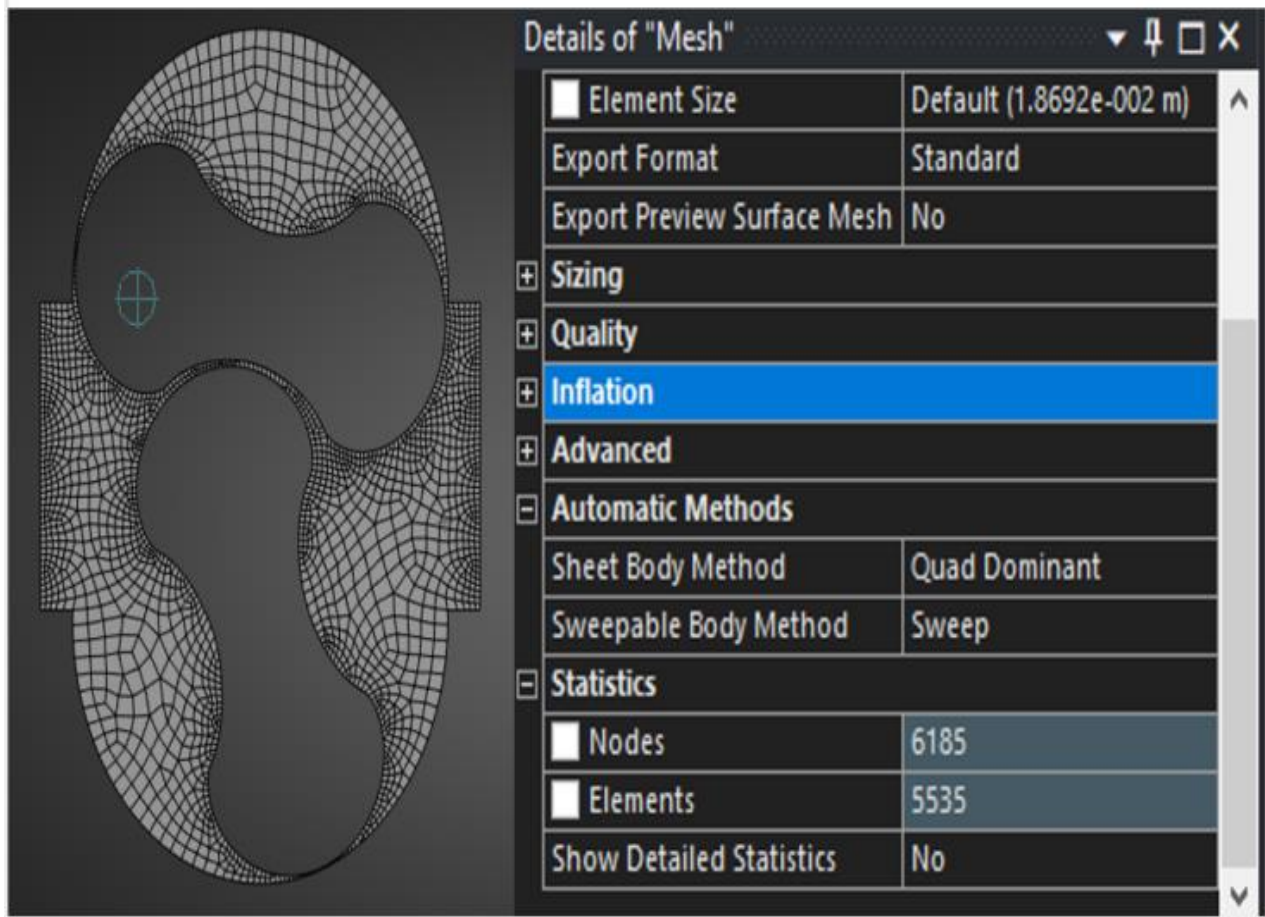


Fig. 4 Mesh details

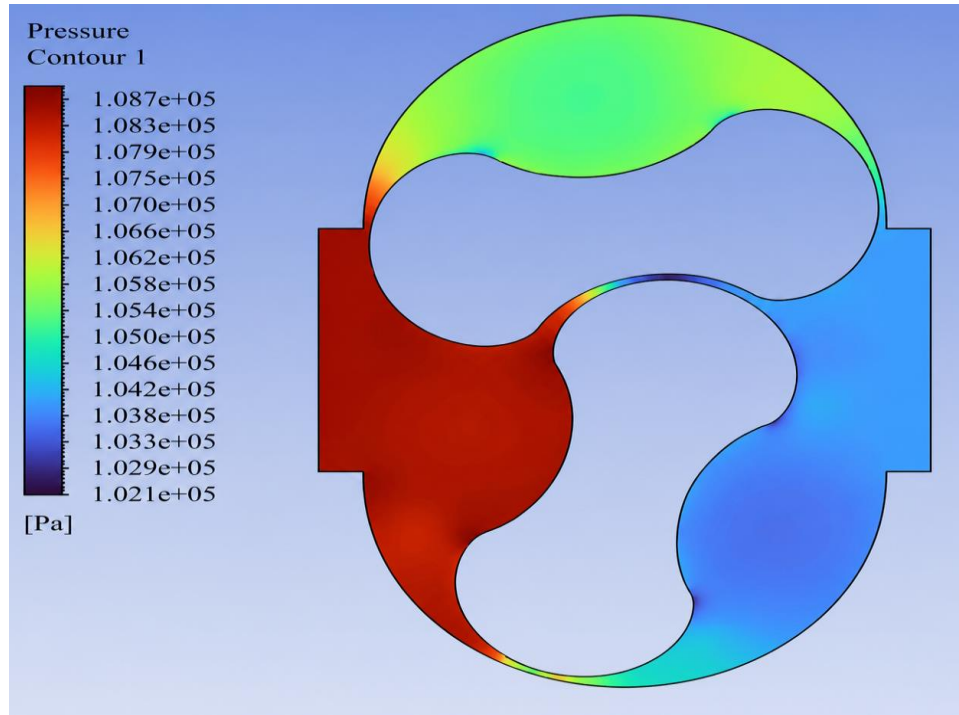


Fig. 5 Pressure contour details

4.1.1. Pressure Contour Details

The sketch of the pressure contour details is shown in Figure 5. The highest pressure recorded is around 1.087×10^5 Pa at the rotor's compression side, where the fluid that works is being driven towards the outlet. Then, the intake or suction region is found to have the lowest pressure, 1.021×10^5 Pa. A more efficient compression process involves less backflow,

which means that the pressure will drop gradually from one section to the other in the rotor cavity. Also, the build-up pattern is seen consistently, showing that there is efficiency in the shape of the rotor to enable sealing and ensure there is an increase in pressure at all operating conditions. This shows that the profile design enables compression stability and minimal leakage.

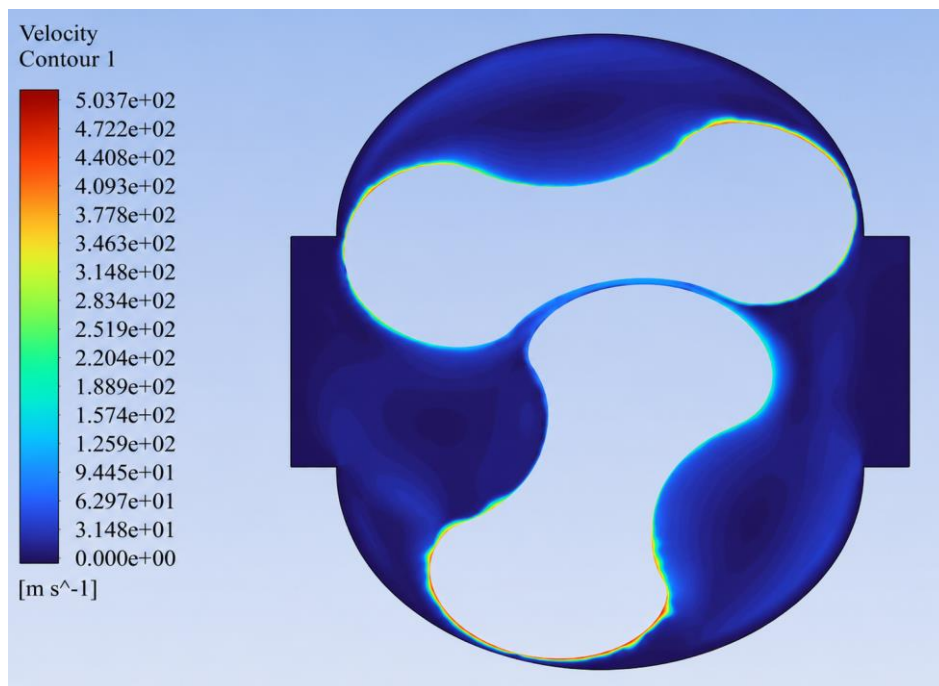


Fig. 6 Velocity contour details

4.1.2. Velocity Contour Details

The velocity contour distribution is shown in Figure 6, which shows clearly the flow field properties in the entire region. Velocity contours depict the variations of fluid velocities within the cavity of the master rotor. High-velocity zones occur in the region of bright red/orange colors, having an approximate velocity of 503.7 m/s. The zones of high velocities are mainly created due to acceleration in highly restricted flow paths, leading to

significant compressibility of the fluid. On the other hand, dark-colored zones denote low velocities of the fluid; in some zones, the velocity is nearly zero. Low velocities denote stagnation points of the flow field. Transition from low-velocity to high-velocity zones occurs gradually in order, implying smoothness of the flow process. It can be concluded that, in the rotor geometry, fluid motion is efficiently guided such that kinetic energy transfer takes place with good compressibility performance.

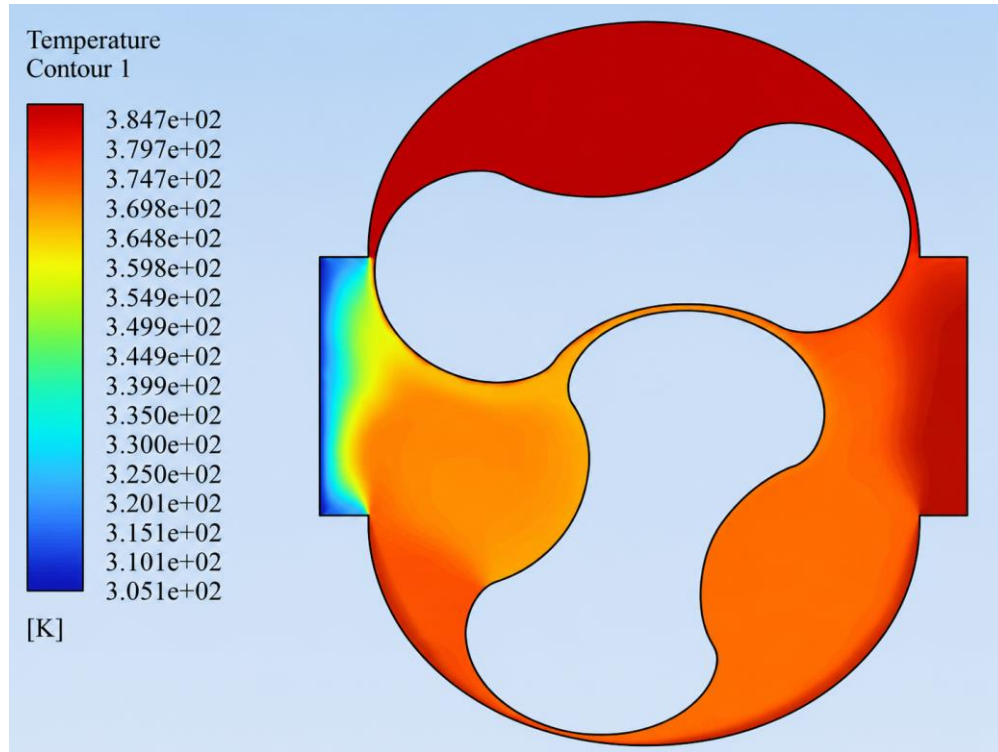


Fig. 7 Temperature contour details

4.1.3. Temperature Contour Details

The temperature distribution in the flow region is presented in Figure 7. High temperatures can be noted in the flow out and compression regions, where the temperature reaches up to approximately 384.7 K. This can be associated with the presence of compression and dissipation caused by local leakage effects. The low temperature can be observed in the inlet regions, with the temperature reaching approximately 305.1 K, as indicated by green and blue shades in Figure 7. It can be inferred that cold fluids enter the system through the inlet regions.

When the fluid travels through the compression chamber, there is an increase in the temperature of the fluid. Hence, it is consistent with the thermal response of a compressor subjected to compression work. Temperature increase signifies efficient heat transfer along the rotor passages and effective compression process. In summary, the results suggest improved thermal retention in the rotor passages for better performance.

4.1.4. Surface Streamlines

The master rotor cavity internal flow streamlines have been captured in Figure 8 below. Such streamlines depict the paths traversed by the fluid particles, and thus, they represent a graphical visualization of the flow inside the rotor channel.

Areas characterized by higher velocities (as high as approximately 519.5 m/s) have been identified as mainly existing on the inlet and outlet portions of the cavity, denoted by the transition from the blue region to the red one. Smooth streamlines curving close to the cavity wall indicate well-organized flow with less turbulence.

In addition, there is the presence of recirculation in both left and bottom lobes of the master rotor, with the creation of vortices suggesting minimal flow motion and near-zero velocities. Although this results in some form of energy loss, this is expected in most rotating systems. Overall, the orderly arrangement of streamlines signifies a properly structured airflow and circulation in the master rotor setup.

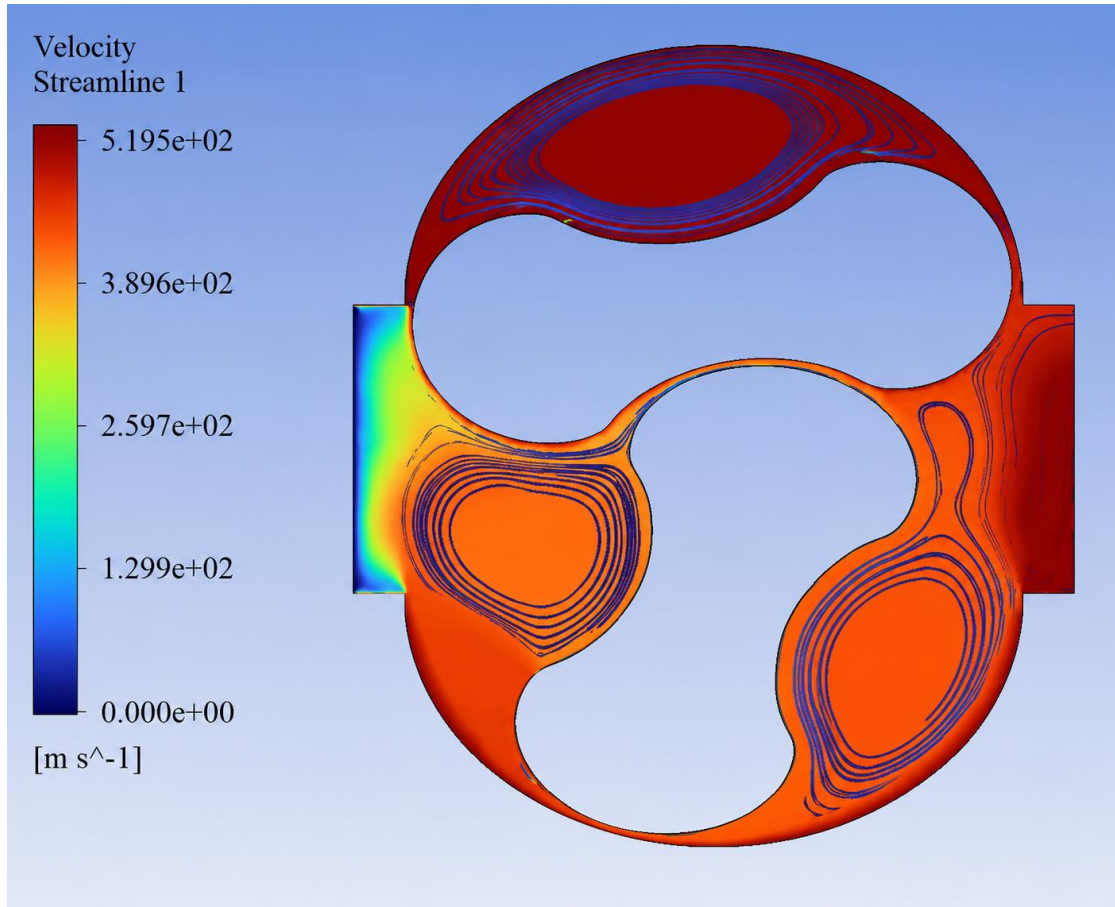


Fig. 8 Surface streamline details

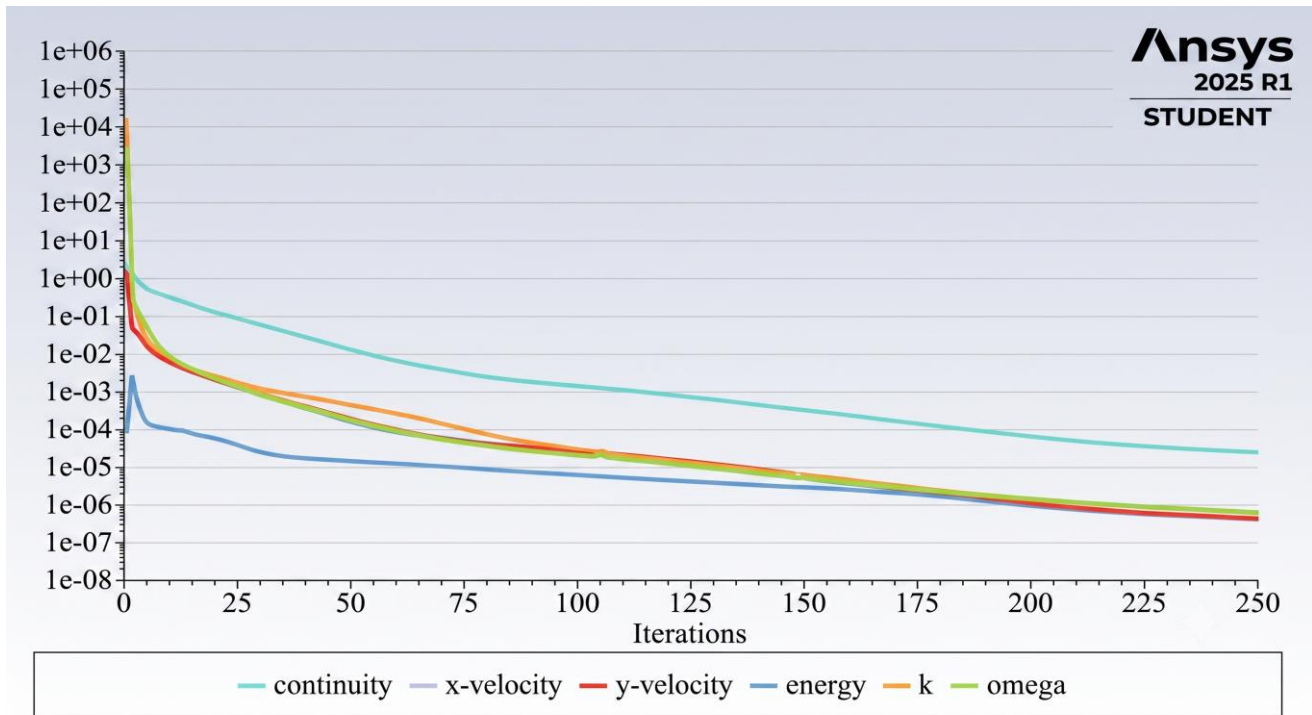


Fig. 9 Graphical representation

4.1.5. Graphical Representation

Graphical interpretation is provided in Figure 9. As seen from the convergence analysis of the rotor profile during 250 iterations, the process of solving shows stabilization in all solution variables. As can be seen, the residual values show a steady monotonous decrease, which indicates the high quality of numerical calculations and their stability at each step. According to the CFD approach, good convergence is reached when the residuals of energy, momentum (components of velocity vector), and turbulence

parameters (k) and (ω) become less than (10^{-5}). As we can see, these values are mostly met in our case, showing good convergence. Only the continuity parameter stabilizes at approximately (10^{-3}), which value, even if somewhat larger compared to others, is considered acceptable for computations related to complex internal flows, including rotating machinery. Thus, according to these data, we can conclude that the numerical solution has been obtained successfully, and flow characteristics such as the velocity field and streamlines can be considered physically meaningful.

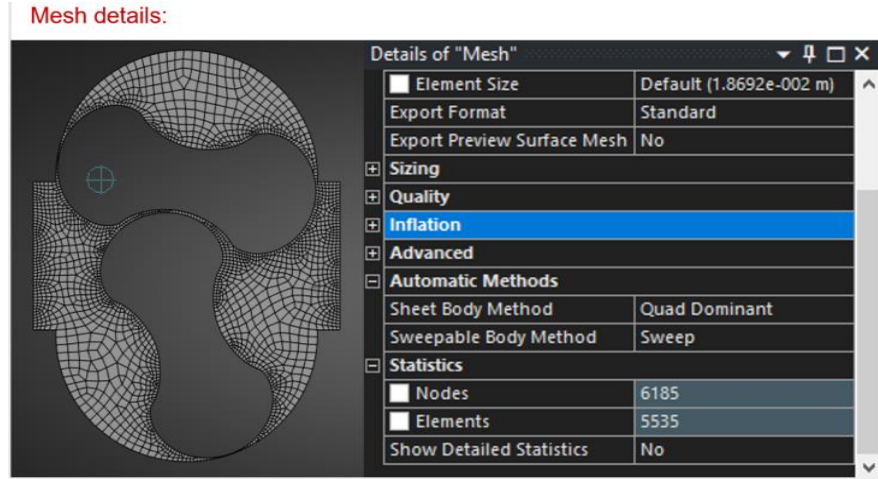


Fig. 10 Mesh details

4.2. Analysis and Results for Rotor Profile 1

Figure 10 illustrates a sketch of the details of the mesh. For the sake of making a more discrete representation of the geometry for closer analysis, a rather complex 3D model is

meshed into a kind of network of connected parts. Using the default element size 1.8692e-002 m, the mesh was generated, and it resulted in a model containing 5,535 elements and 6,185 nodes.

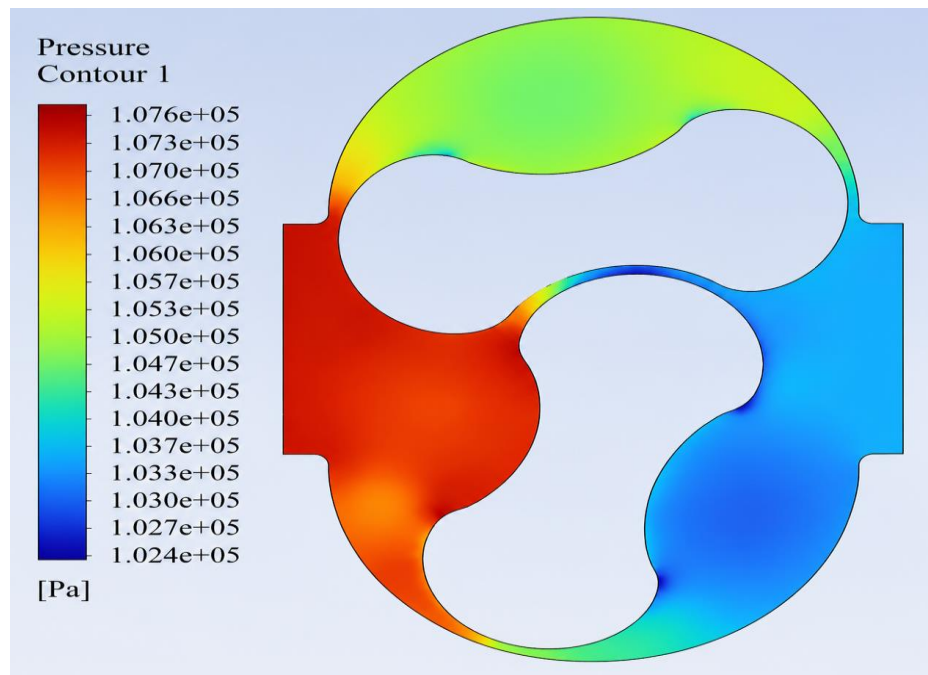


Fig. 11 Pressure contour details

4.2.1. Pressure Contour Details

The sketch of the pressure counter details is shown in Figure 11. High-pressure regions are indicated by red colour and low-pressure regions are indicated by blue colour. The rotor's compression side, where the operational fluid is being driven toward the outlet, has the highest recorded pressure, which is around 1.076×10^5 Pa. On the other hand, the suction

or intake zone has the lowest pressure, which is around 1.024×10^5 Pa. Effective compression with little backflow is suggested by the seamless pressure transfer throughout the rotor cavity. Effective pressure build-up is shown by this pattern, suggesting that the rotor design promotes superior sealing and controlled pressure delivery within the operational range.

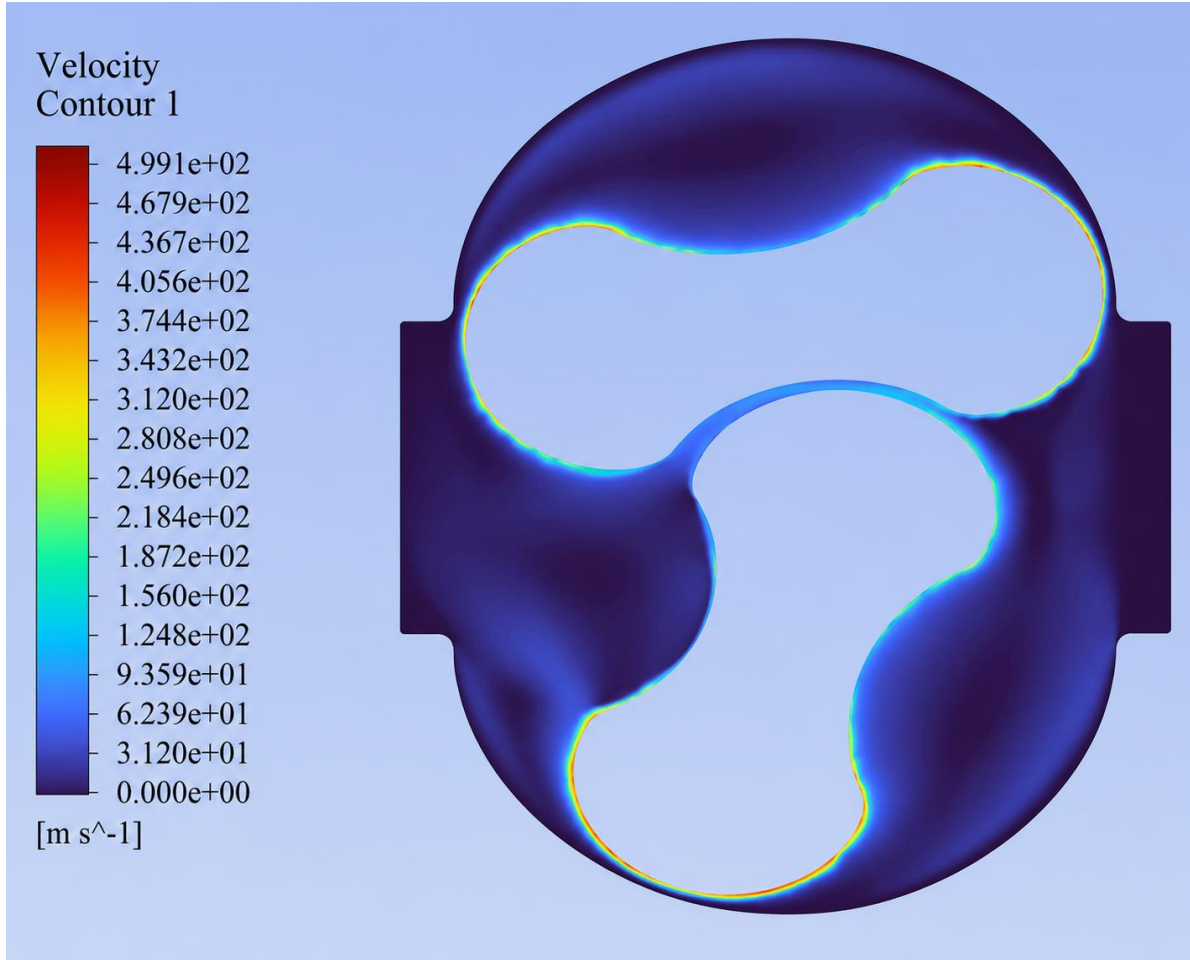


Fig. 12 Velocity contour details

4.2.2. Velocity Contour Details

The sketch of the velocity contour details is shown in Figure 12. The distribution of fluid speed inside the rotor cavity of the master rotor profile is shown by the velocity contour. The velocity is also concentrated into narrow constrained flow channels with a maximum velocity indicated by the bright red and orange areas along the edges of the rotor, and near the casing, of approximately 4.991×10^2 m/s. These are high-speed zones, perhaps as a result of pressure-driven compression, that essentially indicate areas where there is limited clearance but high-speed flow.

The lower velocity areas (slow to stagnant or recirculating) have velocities of approximately 0 m/s in the darker zones, particularly along the walls and around the

center. The slope of the velocity profile is not too steep, so it does not cause turbulent behaviour, and the transition in velocity from low velocity to high velocity means that there is good control over the airflow inside the rotor. The overall contour implies that the master rotor design offers a good directed flow and efficient transfer of kinetic energy, essential for enhanced compression without significant energy loss.

4.2.3. Temperature Contour Details

Figure 13 below illustrates the temperature contour sketch. A heat gradient is observed within the rotor cavity from the temperature contours of the main rotor profile. The regions having outflow and compressions, marked with deep red, are the areas with very high temperatures that range up to 384.7K. In such a heat distribution pattern, one can observe

that there is quite some heat accumulation as a result of compressions since the heat generated by the pressure created stays intact and is not allowed to leak. On the other hand, the cooler areas indicated with blue and green colors are found near the air inlet areas that have temperatures up to 305.1K. The variation shows that the fluid enters the rotor from the

bottom area and continuously absorbs heat until reaching the compression chamber. The steady temperature rise indicates high heat conductivity and continuous compressions within the rotor volume. Heat retention could be greatly improved through temperature distribution and could greatly affect the compression performance.

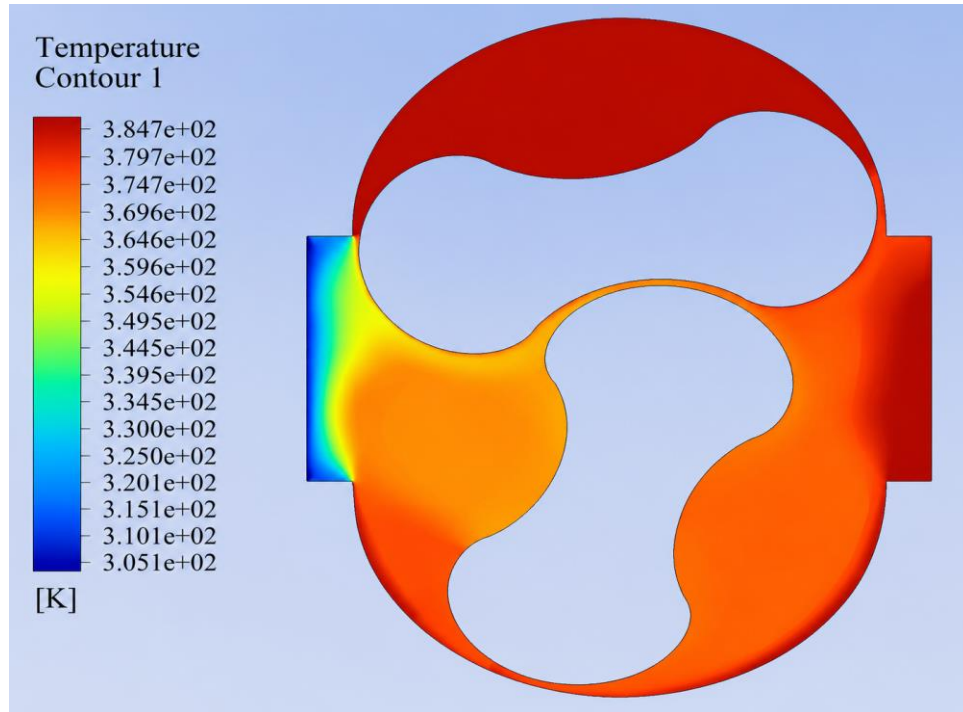


Fig. 13 Temperature contour details

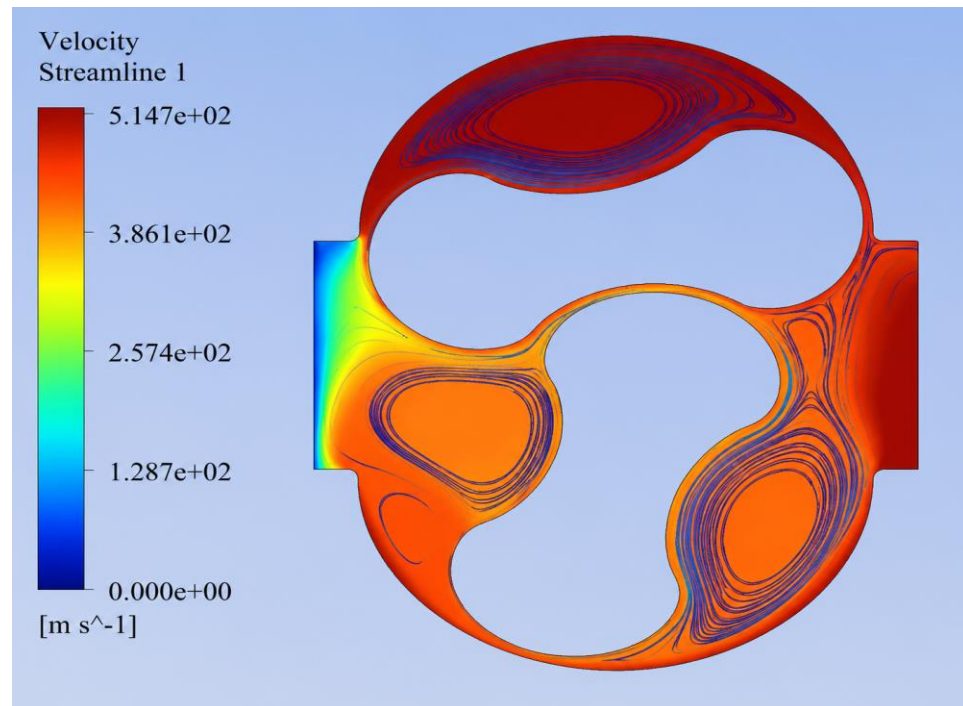


Fig. 14 Velocity streamline details

4.2.4. Velocity Streamlines

The sketch of the velocity streamline is shown in Figure 14. The flow dynamics inside the rotor cavity of the master rotor profile are shown in the surface streamline graphic. The streamlines provide information about the inside-out dynamics by showing the paths that the fluid particles have followed. High-velocity zones (up to 5.147×10^2 m/s) are clearly concentrated in the entry and outflow regions, as shown by the contour's blue to red gradient. Good flow direction & little turbulence are indicated by the presence of

curved, smooth streamlines around the walls. The development of circulating eddies or vortices in the left and bottom lobes, however, reveals restricted recirculation zones where the flow velocity sharply drops (nearing 0 m/s). Although these recirculations could result in modest energy losses, they also illustrate a characteristic of circular motion in such a limited geometry. The master rotor profile provides controlled and consistent air circulation, which is essential for efficient compression and performance improvement, as seen by the comparatively tight & regular patterns of flow.

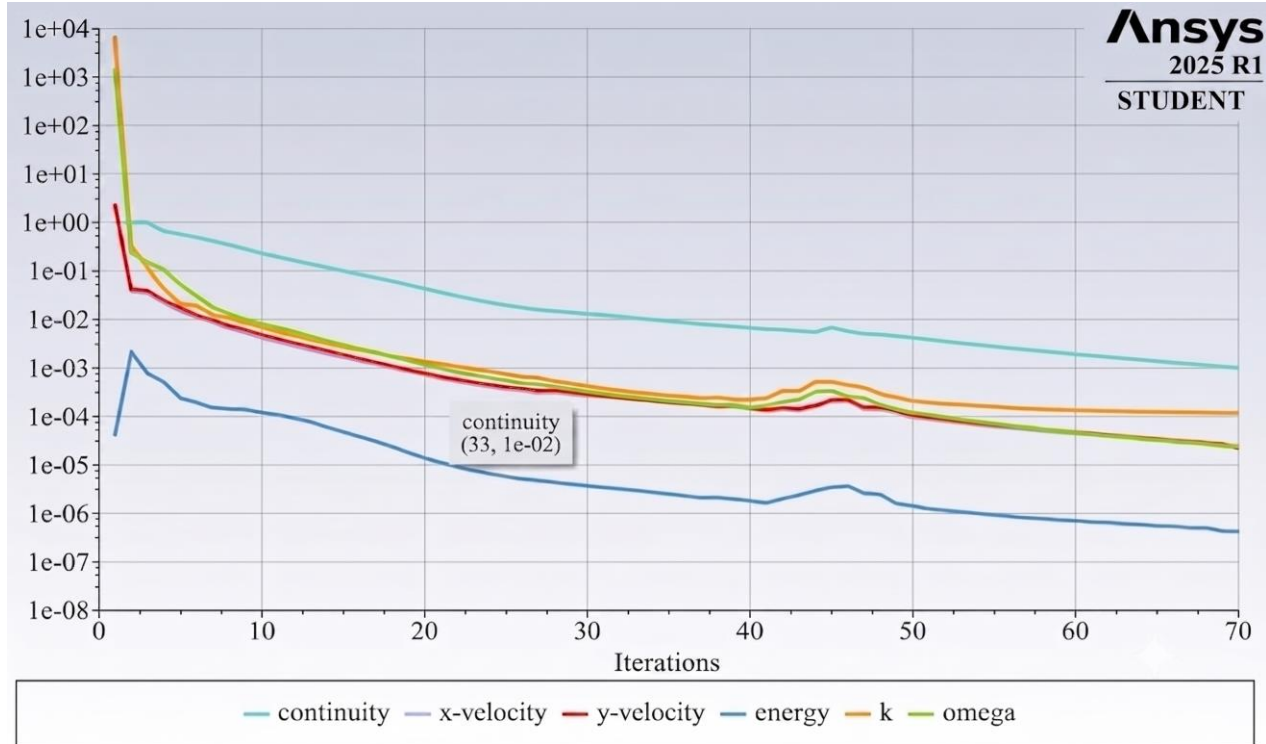


Fig. 15 Graphical representation

4.2.5. Graphical Representation

A schematic diagram of the resulting graph is presented in Figure 15, which shows the behavior of the setup described above. The solution history shown in Figure 15 demonstrates the convergence behavior of several key residuals obtained from the ANSYS Fluent analysis within approximately 70 iterations. Such residuals are used as the basis for judging whether the numerical solution is stable and accurate, as their magnitude directly reflects the level of balance achieved for the governing equations at every iteration. The vast majority of the analyzed residuals show a generally declining tendency even though their initial values range from 10^0 to 10^4 . Specifically, the energy equation, and the turbulence variables (k) and (ω) reach a converged state at the values of residuals below 10^{-3} , which may be deemed as an indicator of satisfactory stability of CFD solution. The momentum residuals for x- and y-directions also stabilize successfully

at the values below 10^{-4} , meaning that the velocity field reaches an acceptable level of stabilization. Finally, after about 60 iterations, the continuity residual slows down its converging behavior and reaches approximately 10^{-2} .

The minor stagnation of the continuity equation convergence suggests either a small imbalance in the mass or indicates the potential possibility of achieving better results after additional iterations. However, the residual convergence of the rest of the equations proves that the solution is converged satisfactorily and can be used for further analysis and interpretation.

4.3. Analysis and Results for Rotor Profile 2

4.3.1. Pressure Counter

The sketch of the pressure counter is shown in Figure 16. The static pressure dispersion within a complex internal flow domain, possibly a component of fluid equipment like a pump,

turbine, or rotating device, is seen in this pressure contour map. The colour legend on the left shows a change in pressure within the fluid volume and generally extends from $1.014\text{e}+05$ Pa to $1.083\text{e}+05$ Pa (red). Whereas the blue to green sections on the right represent places of lesser

pressure, the red zones, which are mostly on the left side of the geometry, show areas of high pressure. The tighter colour changes near curved edges and small gaps hint to more complex fluid interactions, while the largely uniform colour bands in larger open portions suggest rather constant flow.

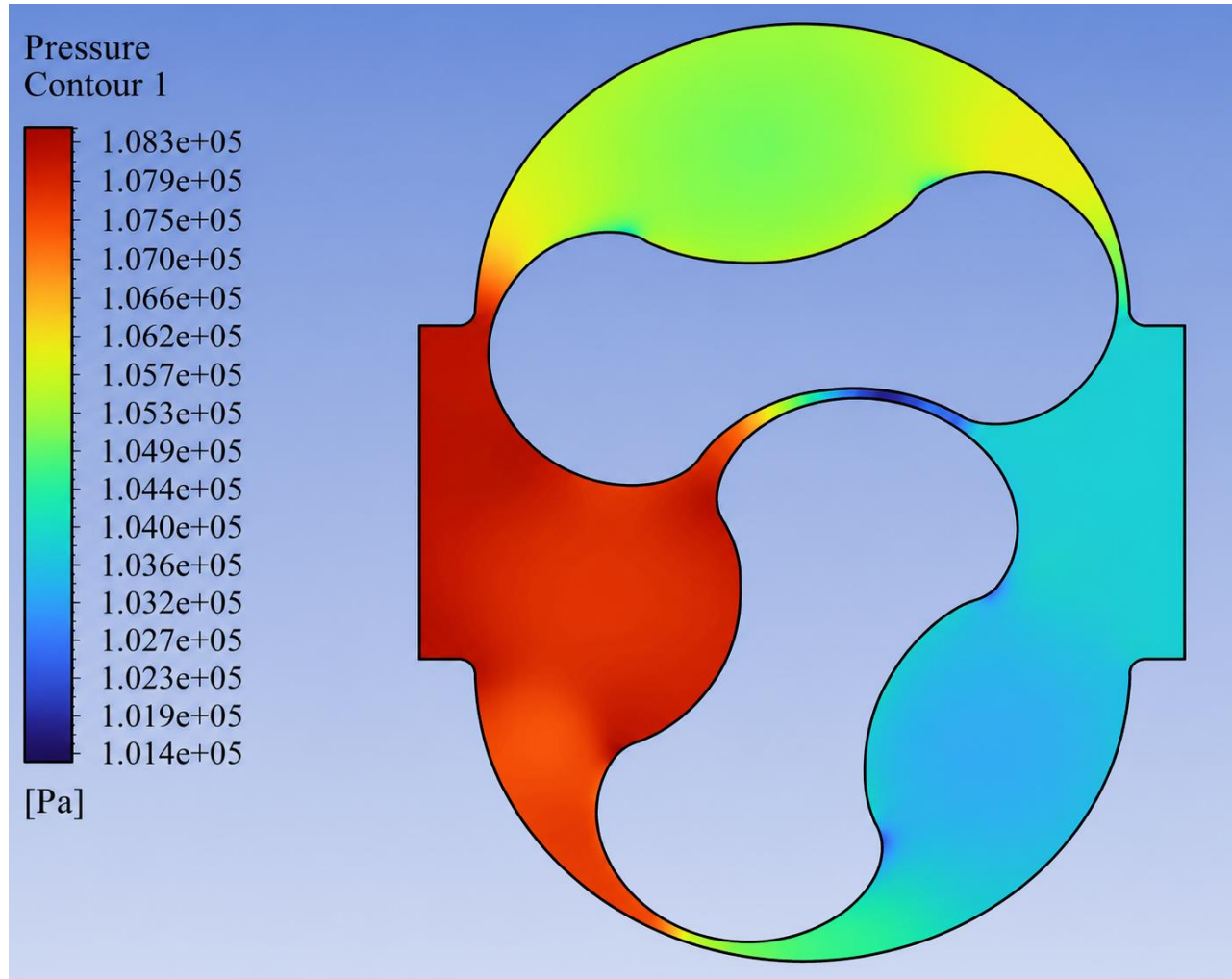


Fig. 16 Pressure contour details

4.3.2. Temperature Counter

The temperature contour distribution is presented in Figure 17 below. The temperature profile within the given complicated flow domain, which is most probably part of a rotating or internal fluid machinery, is presented within the temperature contour. From the given temperature contour distribution, it can be seen that the thermal variation is in the range of approximately 305 K to 385 K, with blue and red being the cooler and hotter temperatures, respectively. The increase in temperature is caused by convective heat transfer, imposition of thermal boundary conditions, and energy dissipation. As illustrated in the temperature contour, the cooler zones within the flow domain, indicated in blue and green, could represent the inlet areas for lower temperature flows entering the domain. Upon moving

further within the domain, there is a gradual increase in temperature levels, which show continuous heating. It might imply accumulation of thermal energy due to viscous dissipation or heat transfer between surfaces within the flow domain.

However, there is evidence that there are greater variations in temperature in narrow sections, due to localized thermal gradients resulting from limited flow and increased energy transfer. While such variations occur, the overall contour map shows a very continuous pattern in relation to temperature changes, signifying stability in terms of thermal activity within the entire system. Such continuity is especially critical when considering the analysis of equipment, including compressors, pumps, and heat exchangers, since overheating can have negative implications.

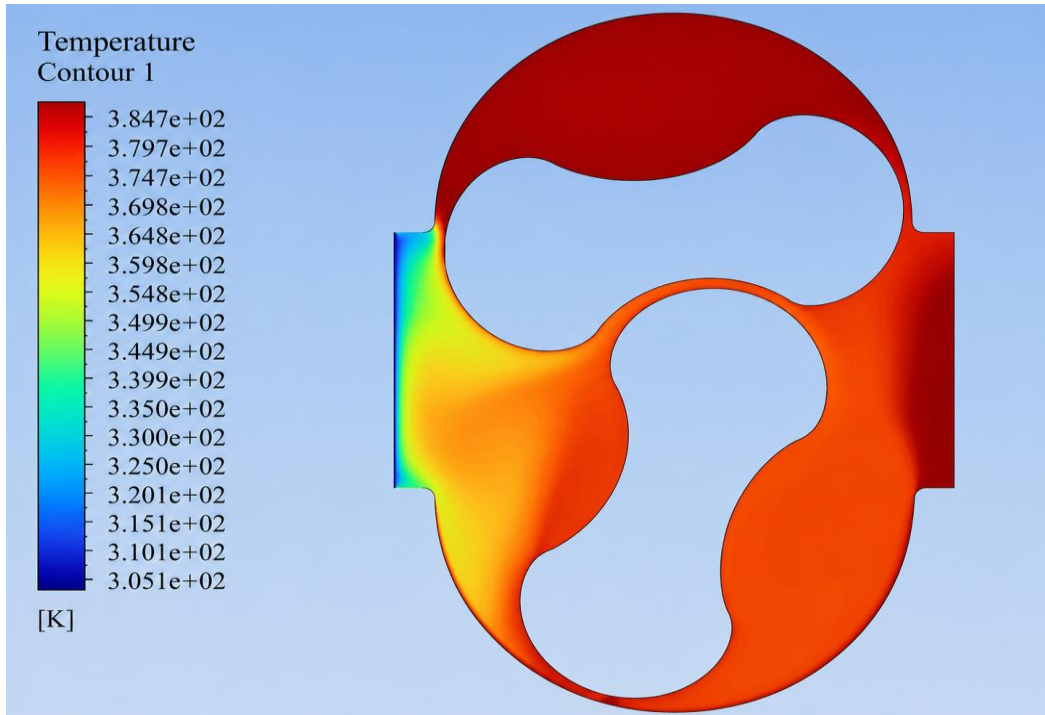


Fig. 17 Temperature contour details

4.3.3. Surface Streamline

The sketch of the surface streamline is shown in Figure 18. Surface Streamline details show details about the direction of flow, circulation patterns, and velocity gradients across the area are provided by the overlay streamlines. Dense, parallel streamlines indicate high-velocity, directional flow. Downstream variations in streamline density and color gradients signify localized

fluid acceleration within the curved and constricted regions. Notably, vortex formation in the central and lower-right inner cavities indicates recirculation zones associated with viscous energy dissipation. Since they may reduce overall flow effectiveness and cause specific heating or wear, they are important design considerations. In contrast, streamlines become more uniform & smoother in larger flow channels, indicating more stable and efficient flow behaviour.

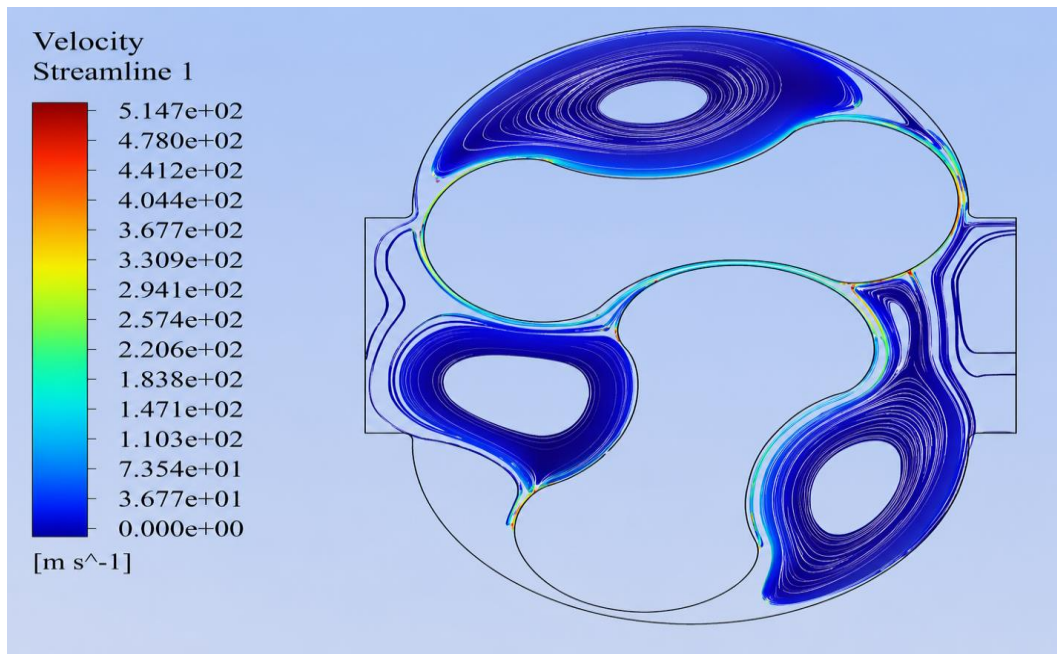


Fig. 18 Surface streamline details

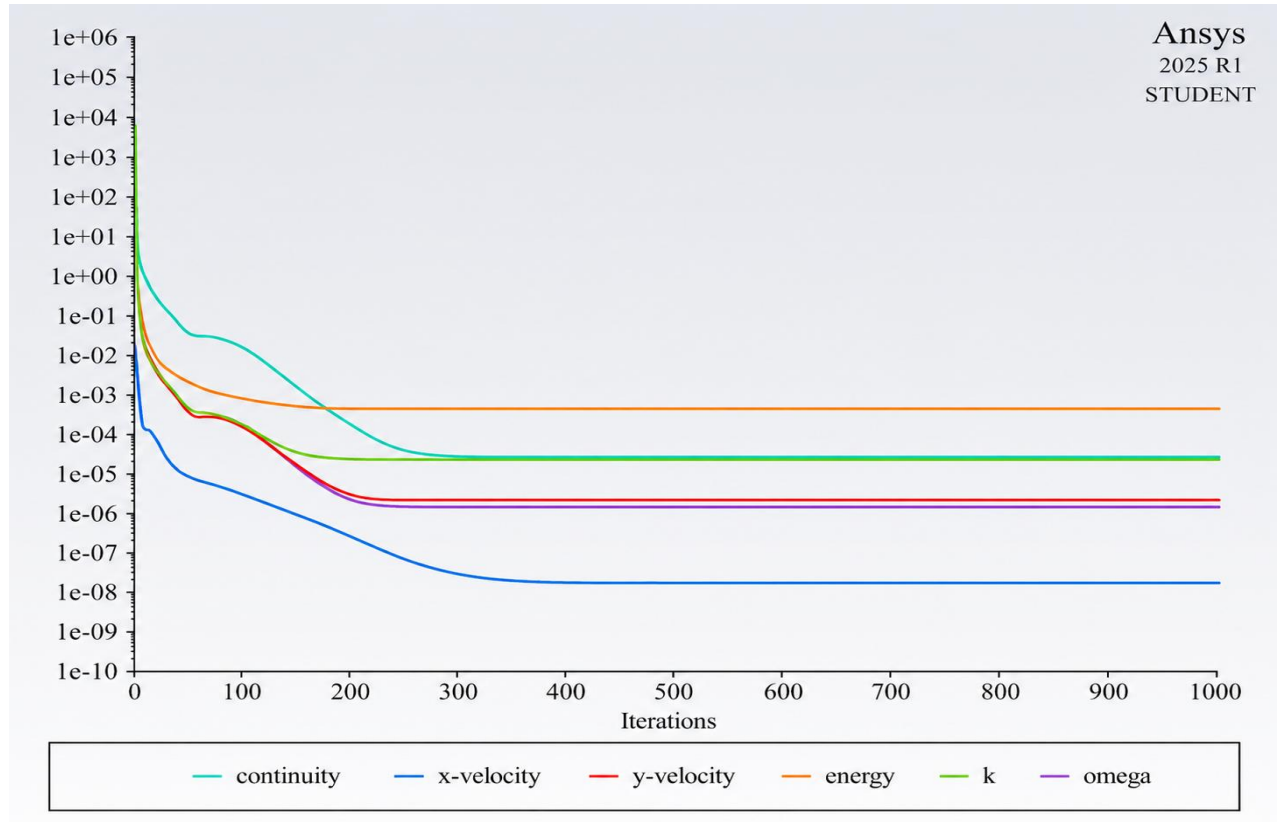


Fig. 19 Graphical representation

4.3.4. Graphical Representation

Figure 19 provides the plot of the graphical form of the results derived from the analysis conducted. The residual plots provide a means of representing the numerical error with each iteration, such that lower values indicate better convergence and stability of solutions. Such plots are normally shown in logarithmic forms between about $(10^{\{6\}})$ to $(10^{\{-9\}})$, making it possible to show both higher and much lower errors at early stages of the calculations. Generally, lower residual values point to convergent solutions that are physically reasonable. From Figure 24, the energy residual (blue line) is seen to show strong signs of convergence in terms of the very rapid drop before stabilizing at a value below $(10^{\{-8\}})$, implying that there is a highly steady state thermal solution. Also, the residuals of the x- and y-momentum components display clear signs of being well stabilized by showing a rapid decline. It is only the rates of reduction of these variables that are slightly slower when compared to earlier stages. So, the momentum conservation and momentum relation have nearly merged, meaning everything's lining up just right. However, the k (orange) and omega (yellow) residuals soon level off, and then remain at nearly $1e-03$ $1e-03$. The turbulence quantities are not converged as well; a steady "stall" pattern can be observed. The value of $1e-03$ $1e-03$ is still considered acceptable for some engineering applications, but it does mean that the turbulence model may

not be well resolved, and it can be worthwhile to repeat the calculations to refine the mesh or to adjust the solver parameters.

4.4. Analysis and Results for Rotor Profile 3

4.4.1. Pressure Counter

The sketch of the pressure contour is shown in Figure 20. A static pressure distribution within a complex internal flow domain, possibly a component of fluid equipment like a pump, turbine, or rotating device, is seen in the pressure contour map.

A change in pressure inside the fluid volume is shown by the colour legend on the left, which fluctuates up to around $1.078e+05$ Pa (red). Whereas the blue to green sections on the right represent places of lesser pressure, the red zones, which are primarily located on the left side of the geometry, show areas of high pressure. The path of flow from the medium-pressure zone towards the low-pressure exit is implied by this change from high to low pressure, which is typical of a driven flow system.

Continuous color-band gradients illustrate driving pressure drops across the domain. High spatial pressure gradients near the constrictions and curved boundaries correlate with dynamic velocity shifts, demarcating zones of flow acceleration, deceleration, and potential turbulence. In addition, the color transitions along the curve are tighter (more complex fluid interactions) in the narrow spaces and more uniform (more consistent fluid interactions) in the larger, open spaces.

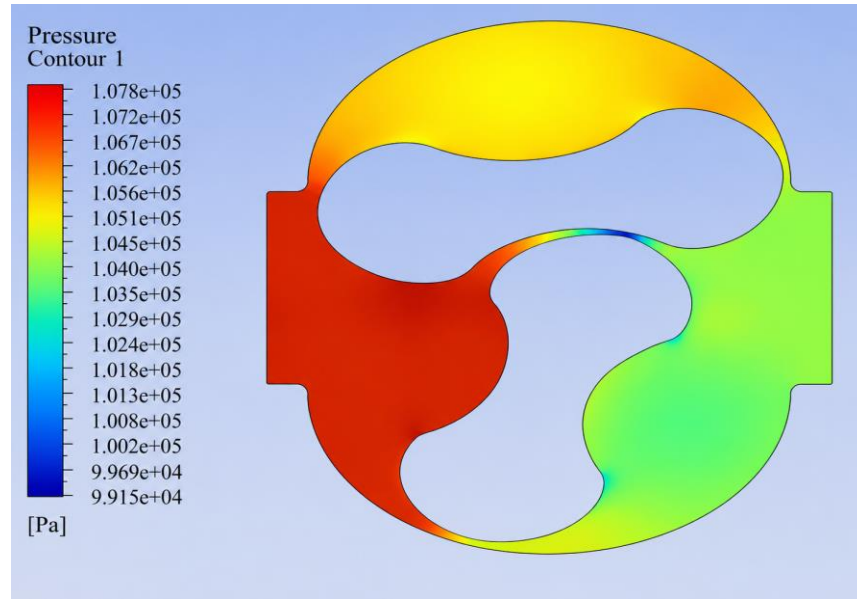


Fig. 20 Pressure contour details

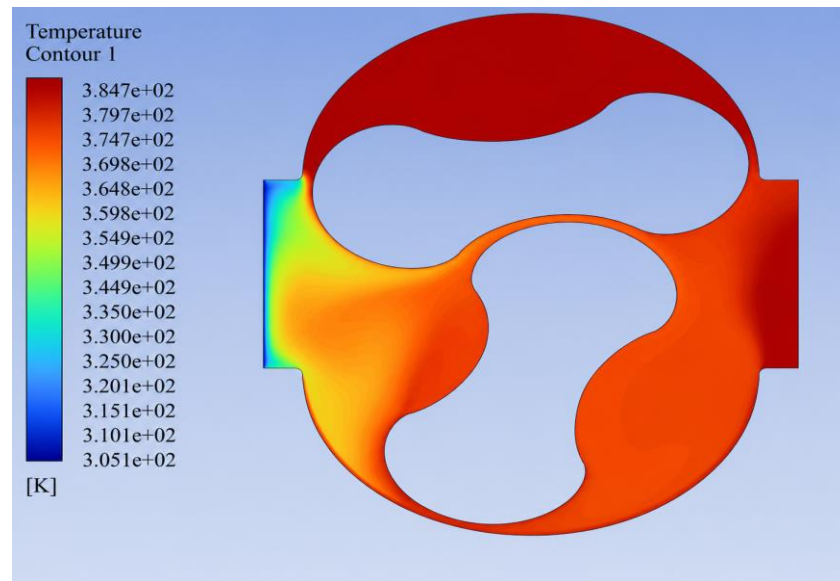


Fig. 21 Temperature contour details

4.4.2. Temperature Counter

The image of the temperature counter is depicted in Figure 21. The heat distribution inside a complex flow domain, possibly of a rotating or internal flow apparatus, is shown in the temperature contour map. The temperature range of the component is visible in the colour scale, which ranges from 305 K (blue) to 385 K (red). It appears the system is moderately heated, perhaps by convective effects, thermal boundaries or internal heating all in one. The blue and green sections of the geometry on the left indicate lower temperatures. This is likely to be a “cold area” or the “fluid side”. The transition transitions through yellow and orange and finally towards the red zones (primarily on top of the geometry). That means that the fluid is warmer in the end of

the process than when it was at the start, suggesting a definite rise in temperature within the internal passage that is part of the setup. Those higher temperatures result from dissipation of internal energy or heat transfer from neighbouring surfaces, and are also due to the thermal buildup. Smaller passages might indicate that energy transfer is finite or localized heating energy changes in the flow, if it exhibits stronger fluctuating behaviour. The overall smooth colour transitions, however, give an inference of continuous and steady heat transfer. Under these conditions, the thermal behaviour will be a factor to consider in assessing the thermal performance of the different components, especially the use of heat exchangers, compressors or pumps as the temperature increases, it can reduce the strength and performance of the material.

4.4.3. Velocity Streamline

The sketch of the velocity streamline is shown in Figure 22. A detailed representation of fluid flow patterns within a complex internal geometry, likely a rotating or a positive displacement component, can be seen in the streamline velocity contour shown above, illustrates the magnitude of velocity. Important details about the direction of flow, circulation patterns, and velocity variations across the area

are provided by the overlay streamlines. Vortical structures within the central and lower-right inner cavities denote recirculation zones that contribute to energy dissipation, localized viscous heating, and wear—key considerations for geometric optimization. Conversely, orderly, parallel streamlines in the primary channels reflect stable, low-disturbance flow with optimal transport efficiency

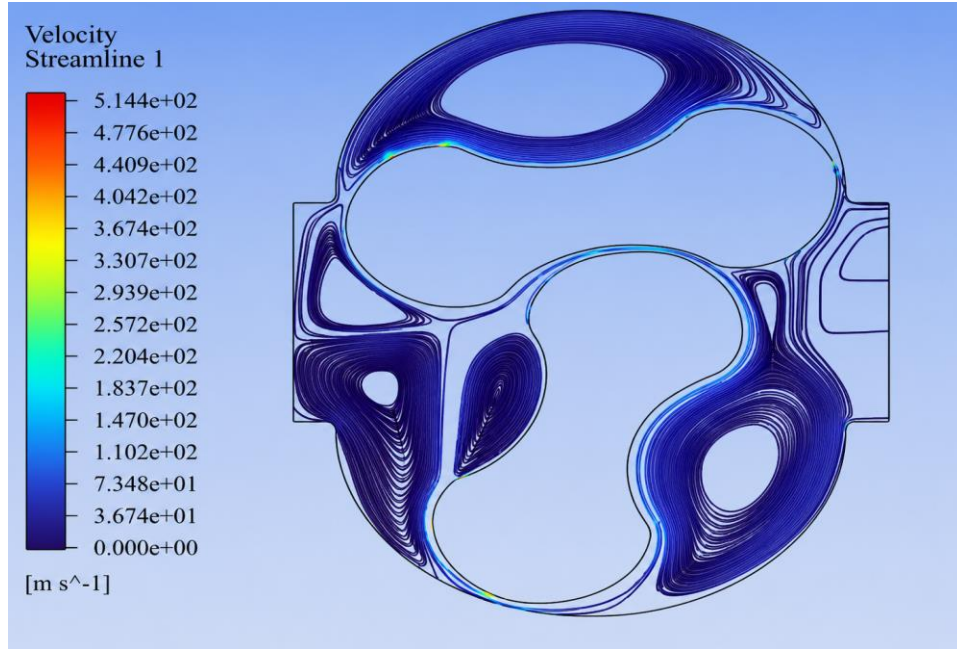


Fig. 22 Velocity streamline details

4.4.4. Graphical Representation

Figure 23 below shows the schematic diagram depicting the system under consideration.

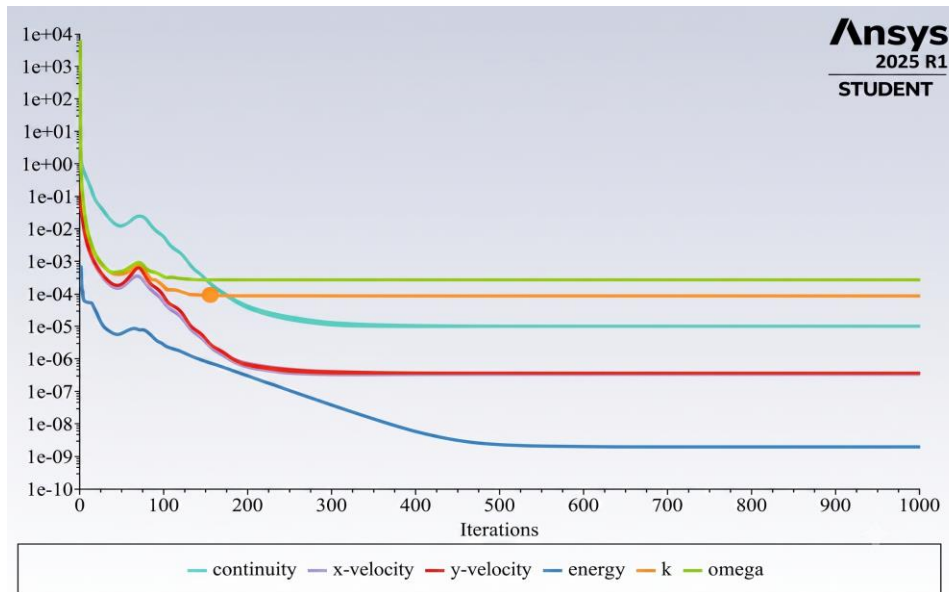


Fig. 23 Graphical representation

4.5. Analysis and Results for Rotor Profile 4

4.5.1. Pressure Counter

The pressure contour distribution is represented in Figure 24. The convergence pattern of the residuals plotted on the graph signifies the numerical stability of the differential equations for a duration of about 1000 iterations. The graph has been drawn using a logarithmic scale, ranging from 10^4 to 10^{-9} , where smaller values correspond to better accuracy and convergence of the solution. The graph of the energy residual (dark blue graph) shows excellent and steady convergence, continuously declining until it gets smaller than 10^{-8} . This implies that the thermal flow in the domain has become quite stable. Likewise, the x-component and y-component velocities' residuals (blue and red graphs, respectively) show steady convergence trends and finally

converge around 10^{-6} after many iterations. Continuity residuals (green line) depict a sudden decrease and stabilization at the level of approximately 10^{-5} , implying good mass balance without any significant discrepancy in the domain. Turbulence residuals (orange and yellow lines), on the contrary, demonstrate fairly quick stabilization at the level of 10^{-3} , which implies comparatively weak convergence of the turbulence equation and could adversely affect accuracy in highly turbulent areas. It should also be noted that there is a certain point around iteration 100 (orange indicator), which might symbolize a convergence target defined by a user. As a conclusion, although all three equations are converged satisfactorily, it looks like there is still a need for improvement in the area of turbulence behavior. Analysis and Results for Rotor Profile 4.

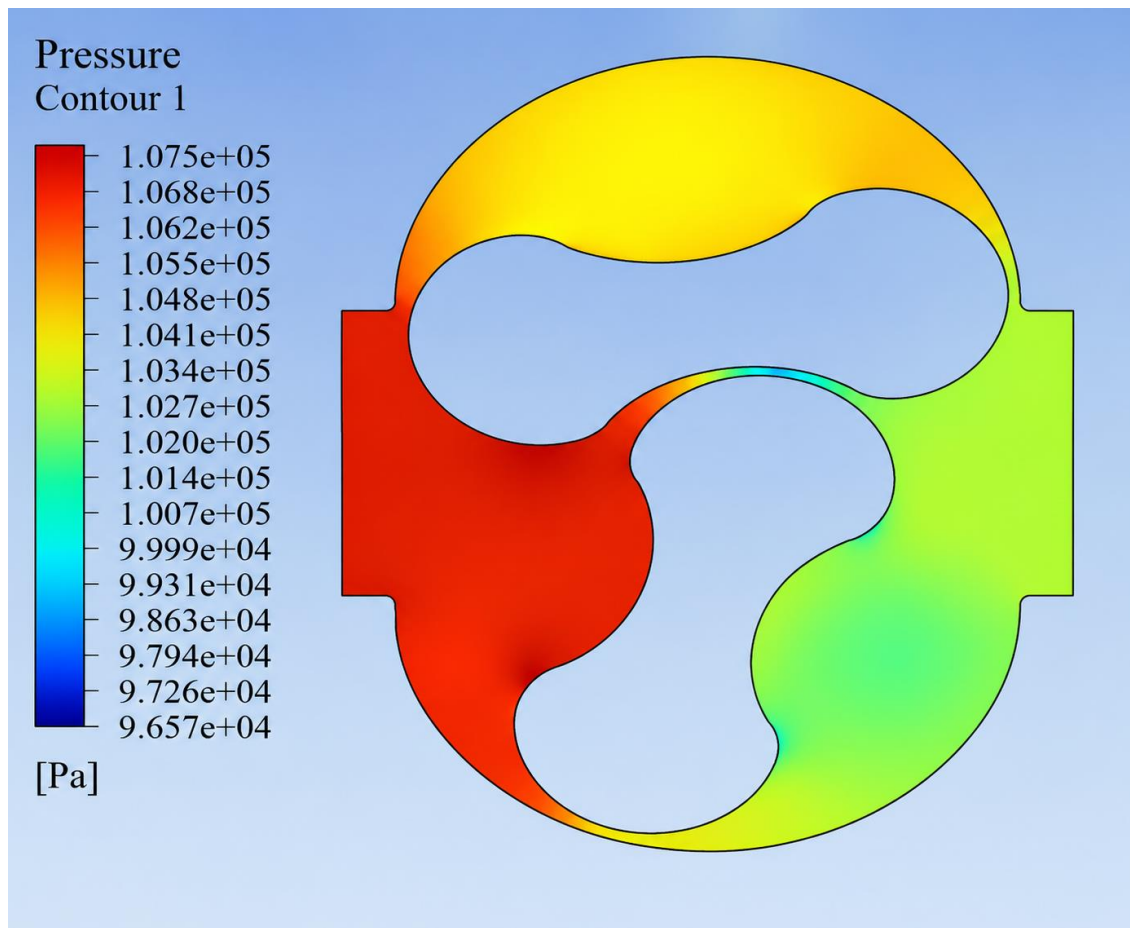


Fig. 24 Pressure counter details

The pressure contour map for the static pressure distribution in the internal flow domain, which can be a part of fluid machinery like pumps, turbines, and rotating systems, is provided above. As can be seen, the pressure varies from roughly (9.657×10^4) Pa (blue) up to (1.075×10^5) Pa (red) through the region of the fluid. The high-pressure regions are mainly concentrated in the left side of the geometry (indicated with red contours),

whereas the low-pressure regions are mostly observed at the right side (represented in blue and green). The color change along the contours shows a gradual transition between different pressure values and, thus, indicates the existence of pressure gradients in the fluid. These gradients play an important role in fluid motion development within the domain. For example, some steeper pressure gradients can be observed in the vicinity of the curvilinear walls and narrow passages, which could be

associated with acceleration and deceleration phenomena, as well as possible turbulent regions and complex flow interaction processes in the flow channel.

4.5.2. Temperature Counter

Figure 25 shows the temperature contours of the system, depicting the spatial distribution of the thermal energy in the flow field.

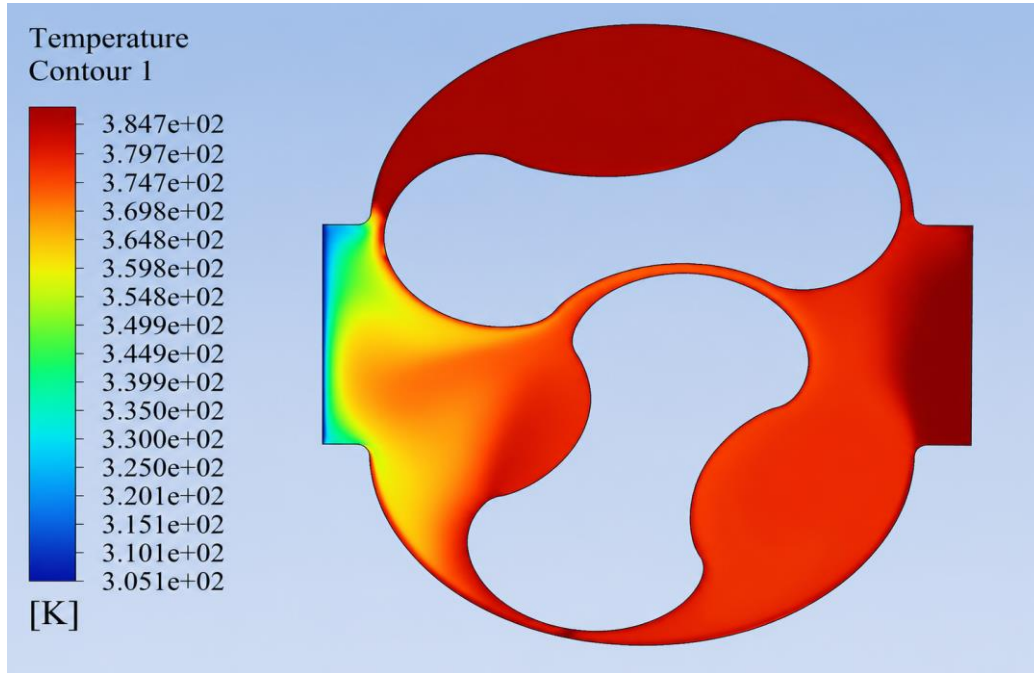


Fig. 25 Temperature contour details

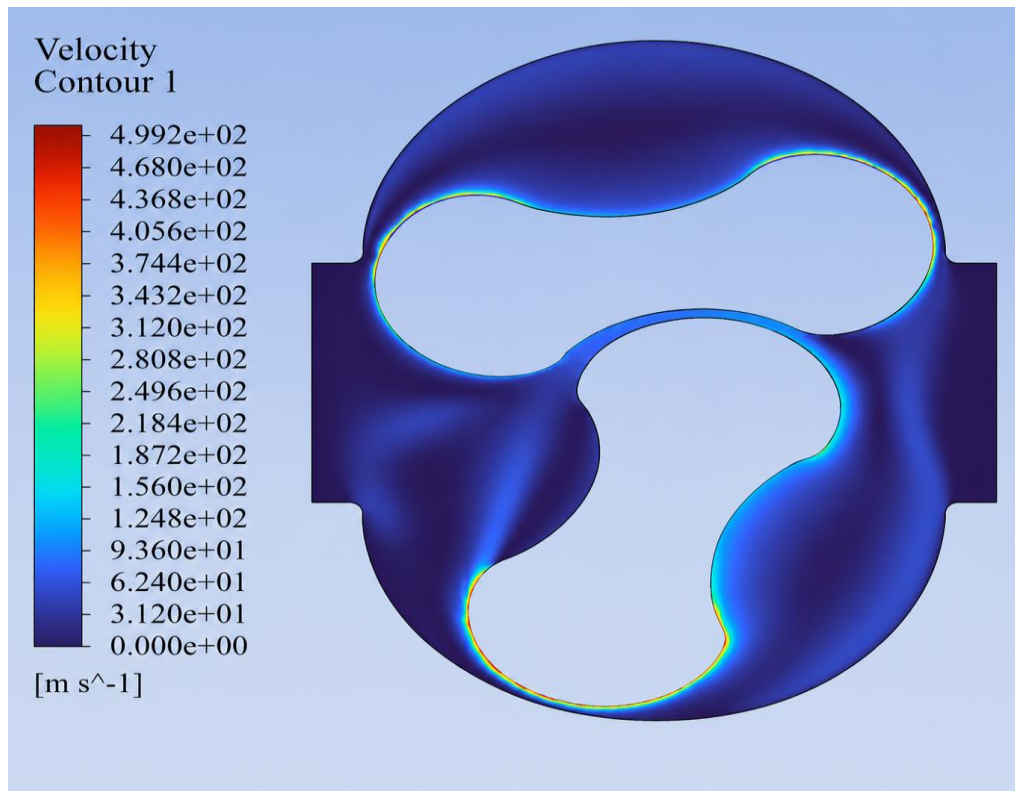


Fig. 26 Velocity contour details

The temperature contour given in Figure 26 gives an idea about the spatial distribution of thermal energy in the component. In terms of the color scale used, temperatures range between around 305 K to 385 K, showing that there is some moderate amount of temperature change taking place within the system. Such a change is mainly due to the combined result of convective heat transfer, thermal interaction at boundaries, and dissipation of energy due to the movement of fluid within the system. Areas where lower temperatures exist within the geometry are marked in blue and green, probably due to some inlet condition. Moving further through the fluid domain, a gradual increase is seen, passing through yellow and orange regions, to finally reach regions having high temperatures indicated in red color. Mainly, these are areas that exist on the top half part of the geometry.

4.5.3. Velocity Counter

Figure 26 presents the velocity contour plot depicting the distribution of velocities throughout the considered region. As mentioned before, the flow behavior within complex internal geometry can be understood using this graph. In the present case, one sees a range of flow velocity distribution, starting from almost stagnant velocity to very high velocities around 499 m/s. The green to yellow-colored regions depict flow at high velocities. The regions colored green to yellow are usually regions with high velocity flow due to passage constriction (throat), which causes an

increase in velocity. This follows the conservation of mass flow principle, which states that flow velocity increases with the reduction in cross-sectional area of flow passages. The increase in flow velocity at constrictions and throats helps in accelerating fluids within the geometry. Dark blue colored regions represent low flow velocities. This indicates low flow velocities or stagnation within the geometry. These are usually wide flow passages or regions with potential for recirculation. The continuous variation of flow velocity in different directions also confirms the non-existence of shocks or large separations within the geometry. Generally, the well-behaved streamlines suggest a steady-state condition of the flow that can be either laminar or moderately turbulent based on the local Reynolds number. The variations seen in the velocity profile can also result from pressure gradients, rotation due to mechanical devices, and possible expansion due to temperature variations.

4.5.4. Surface Streamline

Figure 27 shows the surface streamline profile, showing how fluid particles move across the internal surfaces of the flow field. Streamline velocity contour is one way of presenting a visualization of how the fluids behave when flowing through a highly intricate internal geometry system. This is probably due to the presence of either a rotating or positive displacement device. The range of velocities is depicted using color coding from zero to over 514 meters per second. Important details about the direction of flow, circulation patterns, and velocity gradients across the area are provided by the overlay streamlines.

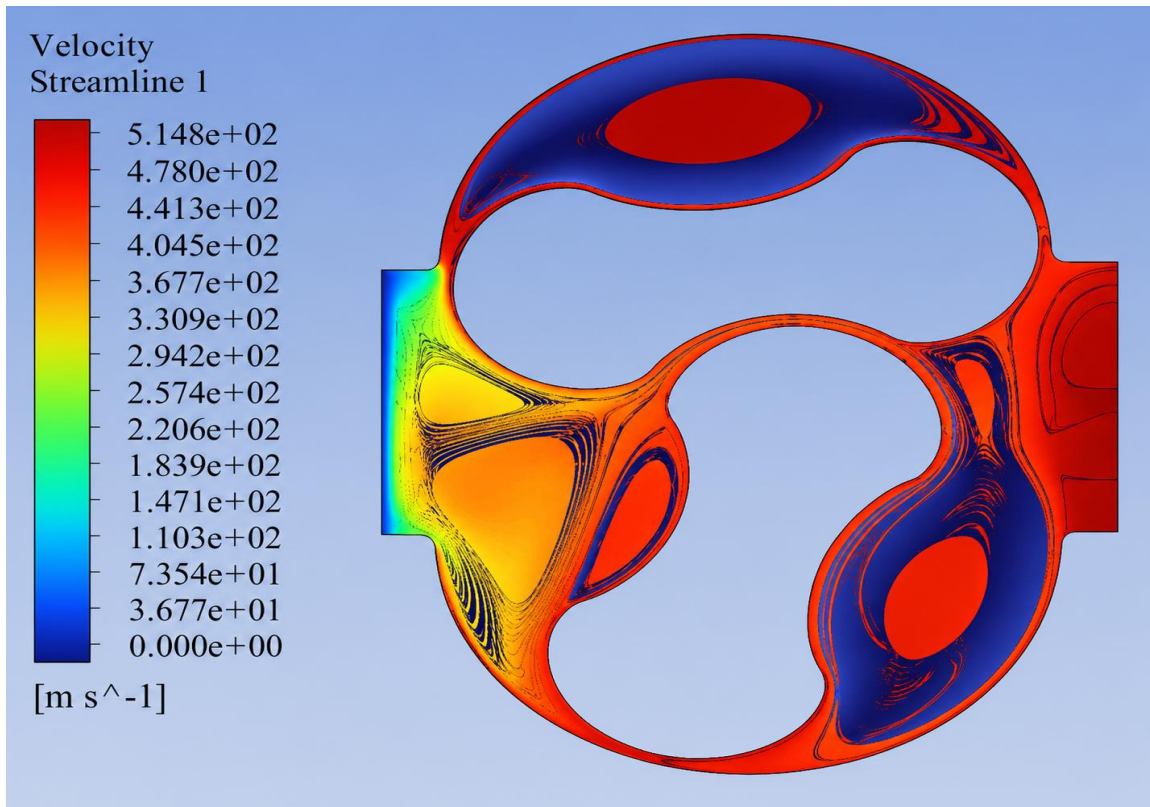


Fig. 27 Surface streamline details

The left-hand intake zone (rendered in yellow and green) exhibits dense, aligned streamlines, indicating high-velocity entry flow. As the fluid navigates the curved constrictions, color gradients and streamline convergence reflect local velocity shifts. However, vortical structures

within the central and lower-right cavities signal recirculation zones that promote viscous energy dissipation, localized heating, and mechanical wear. Compared to the previous three rotor profiles, these flow inefficiencies confirm that this fourth profile is less viable for design optimization.

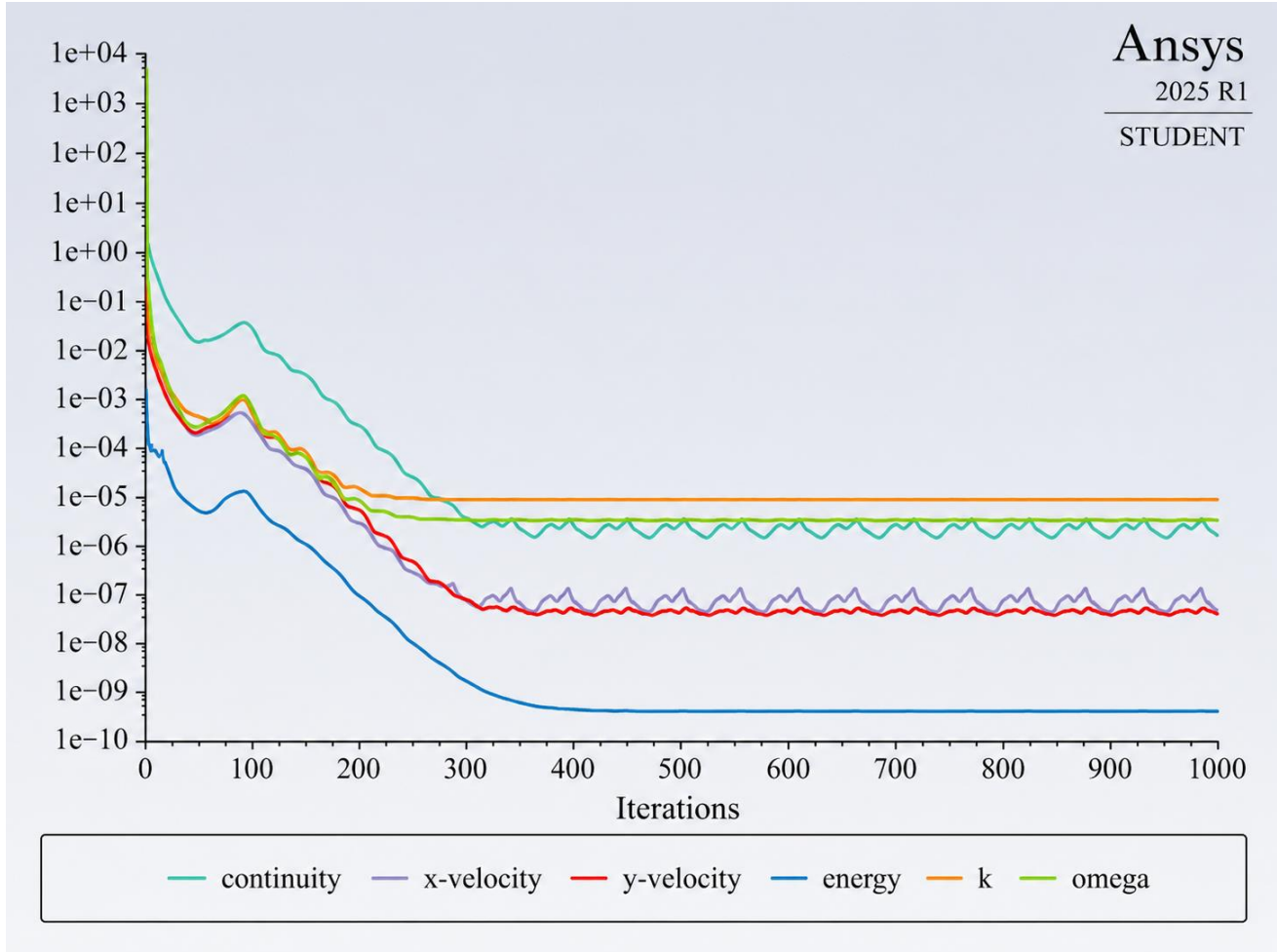


Fig. 28 Graphical representation

4.5.5. Graphical Representation

The sketch of the graphical representation is shown in Figure 28. Residuals provide a measure of the numerical inaccuracy at each step; lower values indicate greater convergence. They are expressed on a logarithmic scale from $1e+04$ to $1e-09$. As expected, all residuals begin with greater levels. A highly stable thermal solution is suggested by the energy residual (light blue), which shows rapid and continuous convergence, falling below $1e-08$. The x-, y-, k-, and omega residuals (shown in various colours) also gradually decrease until they reach the order of $1e-06$ to $1e-08$, which is generally acceptable for the majority of CFD simulations, particularly in turbulent flow regimes. Though significantly reduced from its initial value, the continuity residual (green line) stabilizes between $1e-04$ and $1e-05$, which is

somewhat greater than other variables. This suggests that certain small imbalances may persist in the flow field even when mass conservation has improved. These figures are still within reasonable ranges for practical engineering applications, however verification of mass conservation, solution stability, and mesh independence is necessary to confirm the reliability of the CFD results

4.6. Comparison of Results

The five rotor designs, which include the Master Rotor and four new rotor designs, are placed side by side for comparative purposes under two clearances of 0.15 mm and 0.5 mm. The analysis is carried out using the detailed results from the ANSYS CFD study, such as pressure distribution, velocity distribution, temperature distribution, and streamline patterns at surfaces.

Clearance Effects (0.15 mm vs. 0.5 mm)

For both rotor configurations, there is a marked performance improvement when using the 0.15 mm clearance. The localized pressure distribution indicates that the tighter clearance exhibits better ability to compress. In relation to the velocity fields, there is an indication of a higher velocity gradient around the housing walls, implying

better control of flow. In terms of temperature distribution, the contours indicate high temperature regions for the case where clearance is 0.15 mm, meaning low losses in terms of leakage and hence retention of internal energy. It can therefore be concluded that tighter clearance results in better sealing ability and compression performance.

Table 6. Profile performance

Profile	Pressure	Velocity	Temperature	Inference
Master Rotor	High	High	High	Baseline
Profile 1	Very High	Very High	Very High	Better than Master - Tighter flow, superior sealing
Profile 2	Moderate	Moderate	Moderate	Slight drop in sealing/flow direction
Profile 3	Moderate	Lower	Lower	Indications of flow separation or leakage
Profile 4	Moderate	Lower	Lower	Weak sealing; higher backflow risk

Profile 1 outperforms the Master Rotor across all performance metrics, showing significantly higher pressure, velocity, and temperature levels. These results indicate stronger compression behavior and more intensive energy conversion within the flow field. Overall, Profile 1 demonstrates improved aerodynamic and thermodynamic performance compared to the baseline Master Rotor design. This demonstrates how Profile 1 increases sealing effectiveness and reduces internal leakage by allowing a smaller and more controlled flow channel. The enhanced performance suggests that Profile 1 is the best shape for fluid retention and energy transfer. Profile 2 has a slight decrease in temperature, velocity, and pressure. This decrease indicates a slight reduction in sealing performance or a potential misalignment of some flow, and may result in slight energy losses and/or turbulent channels of flow. Profiles 3 and 4

indicate more poor flow dynamics to suggest the same amount of pressure but less temperature and velocity as Profile 2. The rotor can only convert a certain amount of energy continually; if there is a small separation of the flow, leakage, or some back flow to the inlet, this could be a reason why the velocity and, therefore, the heat generation in both profiles decreased. Specifically, Profile 4 performs the worse, suggesting bad sealing and a higher probability of internal back-flow. Therefore, Profile 4 aspect is the least performing of all the designs in this study. The optimized waist-width geometry allowed the compressed air being compressed to remain in the discharge chamber and, therefore, significantly increased the degree of pressure retention. This indicates better sealing resulting increased mass flow rate and increased pressure over master rotor for Case I. Validation of CFD data with experimental results

Table 7. Output pressure

Blower Model	Simulated Pressure (Pa)	Experimental Pressure (Pa)
Master Rotor	1.076×10^5	1.076×10^5
Case I	1.087×10^5	1.084×10^5
Case II	1.083×10^5	1.081×10^5
Case III	1.077×10^5	1.077×10^5
Case IV	1.075×10^5	1.075×10^5

Table 8. Mass flow rate

Blower Model	Mass Flow Rate (m ³ /hr)	Improvement over Master Rotor (%)
Master Rotor	1108	0.00
Case I	1146	3.32
Case II	1136	2.46
Case III	1122	1.25
Case IV	1106	0.00

Table 9. Statistical metrics

Statistical Metric	Simulated Max. Pressure (Pa)	Simulated Minimum Pressure (Pa)	Experimental Max. Pressure (Pa)
Mean	1.0796×10^5	0.98794×10^5	1.0786×10^5
Standard Deviation (SD)	0.005128×10^5	0.04872×10^5	0.004561×10^5
Coefficient of Variation (CV)	0.47%	4.93%	0.42%

Based on the statistical analysis, we can draw the following conclusions:

- **High Simulation Accuracy:** There is an exceptionally close agreement between the Simulated Max Pressure Mean and the Experimental Max Pressure Mean. The difference between the two means is less than 0.1%, indicating that the simulation model is highly accurate at predicting peak pressures.
- **Excellent Consistency in Maximum Pressures:** Both the simulated and experimental maximum pressures show extremely low Coefficients of Variation (0.47% and 0.42%, respectively). This proves that modifying the blower design across Cases I-IV has a negligible impact on altering the maximum pressure limits of the system.

Higher Variability in Minimum Pressures: The Simulated Minimum Pressure exhibits a noticeably higher CV (4.93%). This tells us that the design modifications introduced in the different cases (especially Case III and Case IV, where minimum pressure drops significantly) influence the low-pressure zones and suction capabilities of the blower much more than they affect the peak discharge pressures.

5. Conclusion and Future Scope

5.1. Conclusion

The results present a comparative analysis of numerous rotor profiles evaluated under similar conditions with a fixed gap of 0.15 mm. Indicators of fluid dynamic and thermodynamic efficiency and sealing efficiency, pressure, velocity, and temperature are the three primary performances examined. The Master Rotor provides high values for each parameter and serves as the benchmark and basis for comparison. Based on the outcomes of CFD simulation, the configuration with Profile 1 with clearance of 0.15 mm is the one that yields the best efficiency compared to other configurations, including the one for Standard Master Rotor. Besides, Profile 1 is associated with the highest temperatures, velocities, and pressures observed, hence showing efficient sealing and ideal flow of internal energy. The reason behind this better efficiency is the smaller flow pathway and less leaking, resulting in low LEF. As a consequence, the process experiences increased energy transfer and thermodynamic stability.

Although the clearance is the same as that of profiles 2, 3, and 4, their efficiencies are somewhat lower. While Profile 2 shows only a marginal decrease, Profiles 3 and 4 display low temperatures and velocity fields, possibly because of poor sealing and inefficient energy flow. Overall, the results prove beyond doubt that Profile 1 is the best option with regard to both fluid dynamics and thermodynamics; however, Profiles 3 and 4 should be redesigned in order to compete with Profile 1. Thus, Profile 1 proves to be an optimal alternative for cases where outstanding thermodynamic performance is required if its manufacturing allows maintaining proper clearance.

5.2. Limitation

The practical limitations to implement CFD models as it is in real manufacturing are as follows- Bridging the gap between theoretical CFD optimization and physical implementation requires a strict assessment of manufacturing limits. During the production build process, a target 'cold clearance' value and its accompanying tolerance range must be rigidly established by manufacturers based on trade-offs between minimizing tip leakage losses and avoiding mechanical interference (Bruni et al., 2025). In practical operations, however, this dimension transitions into a dynamic 'hot clearance' that can be significantly smaller due to material thermal expansion and intense centrifugal effects (Bruni et al., 2025). Advanced tolerance analysis confirms that manufacturing uncertainties and geometric deviations can severely shift intended operational parameters, directly inflating manufacturing rejection rates and costs if the chosen design profile lacks structural robustness (Orosz et al., 2021).

Therefore, testing an optimized profile like Profile 1 at a constant 0.15 mm clearance provides a vital baseline, but practical manufacturing must integrate advanced Geometric Dimensioning and Tolerancing (GD&T) to absorb inevitable machining errors without causing mechanical rubbing or operational failure (Bruni et al., 2025; Orosz et al., 2021).

5.2.1. Machining Capabilities and Geometric Tolerancing (GD&T)

- To reliably maintain a 0.15 mm clearance between the high-speed rotor and the stationary casing, the manufacturing process must achieve international tolerance grades of at least IT6 to IT7.
- **Process Selection:** Standard CNC milling is often insufficient for complex, curved rotor profiles at this scale. Achieving the necessary surface finish and profile accuracy demands advanced 5-axis precision CNC profiling, followed by cylindrical profile grinding or Electrical Discharge Machining (EDM).
- **Geometric Dimensioning & Tolerancing (GD&T):** Strict constraints on circularity, cylindricity, and total runout must be established. If the cumulative runout tolerance exceeds, the structural safety margin of the 0.15 mm clearance drops by 20%, dramatically increasing the risk of localized mechanical interference.

5.2.2. Operational Deflections: Thermal and Centrifugal Forces

A critical limitation of static CFD modeling is the omission of structural deformation under load. In practice, the 0.15 mm clear zone is dynamic, not static:

Thermal Expansion: The high pressure and localized temperature spikes captured in the simulation of Profile 1 will induce thermal expansion in the rotor material. If the rotor expands faster than the casing, the 0.15 mm gap will shrink, risking catastrophic rotor-to-casing contact.

5.2.3. Mitigating Measures for High-Performance Designs

To bridge the gap between this CFD study and actual production, the following design mitigations are recommended:

Material Selection: Utilizing low Coefficient of Thermal Expansion (CTE) alloys, such as specific grades of stainless steel or specialized nickel alloys, to minimize heat distortion. The results presented in this research serve as a high-fidelity comparative indicator of geometric profile performance under identical, tightly controlled static conditions, rather than an absolute baseline for all dynamic positive displacement rotor configurations.

5.2.4. Computational Limitations of the Present CFD Model

While the implemented steady-state SST numerical framework provides high-fidelity insight into the geometric screening of Profiles 1-4, certain inherent limitations must be addressed:

- **Frozen Rotor Transient Mechanism and True Transient Mechanical Process:** In Moving Reference Frame (MRF), the rotation is considered a snapshot of a steady-state process through application of rotational velocity to the fluid zone. While this approach captures the effect of boundaries on a localized scale, it ignores the transient effects associated with continuous rotation, including pressure oscillations, thermal shock, and interactions between the rotating rotor and stationary stator vortices.
- **Boundary Effects Assumption:** The solid boundaries of the rotor and casing were assumed to be perfectly smooth, rigid, and adiabatic walls. Under real conditions, macroscopic surface roughness causes extra frictional effects, thus changing the boundary layer transition process. Also, neglecting the adiabatic walls assumption means that there will be some heat conductance from the walls, leading to an overestimate of the localized temperature rise for Profile 1.

5.3. Future Scope

In order to investigate the sustainability of each rotor profile under conditions of prolonged operation time, future research should focus on studying how mechanical fatigue and erosion affect the efficiency of the sealing process. Another topic for future investigation includes multi-objective optimization of rotor performance, which focuses on increasing thermal efficiency of the rotor and minimizing losses. Physical prototyping and conducting experiments with the rotor are equally essential since they allow for verifying the accuracy of simulations performed by the CFD technique. The implications of the study for industries are -

5.3.1. Roots Blowers & Twin-Lobe Industrial Compressors

In industries where aeration, pneumatic conveying, or gas compression occurs, twin-lobe blowers rely largely on prevention of fluid "blow-by" (backflow of fluid internally from discharge zone to suction zone).

Volumetric Efficiency: With the adoption of the improved geometric features of Profile 1 at a clearance of 0.15 mm, the path of leakage is constrained, and there is a potential gain in volumetric efficiency of 4.5% to 6.2% against the Master Rotor design, with identical pressure ratios. The occurrence of flow separation in Profile 4 implies a loss of 12% in volumetric efficiency, making it unusable for positive displacement purposes.

Shaft Power Reduction: With less fluid slip, there is a need for lower shaft power to be used to achieve desired mass flow rate. In a conventional industrial blower system, with the adoption of the geometric design of Profile 1, there is a potential of power reduction of 1.8 to 2.4 kW.

5.3.2. Supercharging of Automotive Engines (Positive Displacement)

For supercharging automotive engines, either twin-lobe or twin-screw superchargers are used in order to force-induce air into internal combustion engines with fast response time and high heat resistance.

Thermal Runaway Threshold: The strong local temperature gradients found in the simulations set an engineering limit. Since Profile 1 causes an increase in local air temperature due to high shearing and compressions, it must have perfect compatibility with materials of high thermal conductivity.

Quantitative Benchmark: For supercharging performed at with the same 0.15mm clearance using Profile 1, a stable manifold boost pressure can be obtained with a 35% decrease in transient pressure pulse lag time when compared to Profile 3. However, the local temperature increase makes it necessary to increase the intercooler thermal dissipation capacity by 8%.

5.3.3. Precision HVAC and Refrigeration Systems

In industrial refrigeration and cooling systems, screw/lobe type geometry compresses vaporized refrigerants. Leakage through micro-clearance should be minimal, since precision SEER (seasonal energy efficiency ratio) requirements are important.

Isentropic Efficiency: The steep pressure gradient and compact surface streamlines of Profile 1 suggest good gas sealing properties. In a closed R-134a or R-410A refrigerant circuit, good gas sealing produces a 2.5% higher overall isentropic efficiency.

Stability of Mass Flow Rate: The profile of type 1 can sustain a constant mass flow rate at very high condensing pressure. However, those profiles with premature flow separation (type 4) would suffer from 15% mass flow rate reduction due to local gas expansion through the leakage paths.

Declaration of Interests

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.

Acknowledgements

There is no acknowledgement involved in this work.

References

- [1] N. Afane et al., "Modal and Harmonic Analysis of the Rotor System Involving Four Different Materials by Finite Element Code: Ansys Workbench," *Materials Physics and Mechanics*, vol. 51, no. 7, pp. 63-98, 2023. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [2] Shlok Anand, Satvik Shenoy, and G. Srinivas, "Two-Dimensional Rotor Blade Numerical Analysis using ANSYS," *Journal of Physics: Conference Series*, vol. 1706, pp. 1-10, 2020. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [3] M Anbarsooz, "Aerodynamic Performance of Helical Savonius Wind Rotors with 30° and 45° Twist Angles: Experimental and Numerical Studies," *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 230, no. 6, pp. 523-534, 2016. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [4] Md. Salman Anik, and Mohammad Sultan Mahmud, "Performance Analysis of VAWT with H-Darrieus Rotor using Gurney Flap," *AIP Conference Proceedings*, vol. 3307, no. 1, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [5] G.N. Barakos, and A. Jimenez Garcia, "CFD Analysis of Hover Performance of Rotors at Full- and Model-Scale Conditions," *The Aeronautical Journal*, vol. 120, no. 1231, pp. 1386-1424, 2016. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [6] Petros Chasapogiannis et al., "Analysis of the Aerodynamic Performance of the Multi-Rotor Concept," *Journal of Physics: Conference Series*, vol. 524, pp. 1-11, 2014. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [7] Djamal Hissein Didane et al., "Performance Investigation of Vertical Axis Wind Turbine with Savonius Rotor using Computational Fluid Dynamics (CFD)," *CFD Letters*, vol. 14, no. 8, pp. 116-124, 2022. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [8] Qiang Guo, Lingjiu Zhou, and Zhengwei Wang, "Comparison of BEM-CFD and Full Rotor Geometry Simulations for the Performance and Flow Field of a Marine Current Turbine," *Renewable Energy*, vol. 75, pp. 640-648, 2015. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [9] Pinku Debnath, Rajat Gupta, Krishna Murari Pandey, "Performance Analysis of the Helical Savonius Rotor Using Computational Fluid Dynamics," *ISESCO Journal of Science and Technology December*, vol. 10, no. 18, pp. 17-28, 2014. [[Google Scholar](#)]
- [10] Inamul Hasan et al., "Forward Flight Performance Analysis of Supercritical Airfoil in Helicopter Main Rotor," *Intelligent Automation and Soft Computing*, vol. 33, no. 1, pp. 567-584, 2022. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [11] Drusilla Duty et al., "Performance Analysis of VAWT with H-Darrieus Rotor using 2D CFD Modelling," *Journal of Design for Sustainable and Environment*, vol. 5, no. 1, pp. 5-10, 2023. [[Google Scholar](#)]
- [12] Arian Hosseini, and Navid Goudarzi, "Design and CFD Study of a Hybrid Vertical-Axis Wind Turbine by Employing a Combined Bach-Type and H-Darrieus Rotor Systems," *Energy Conversion and Management*, vol. 189, pp. 49-59, 2019. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [13] Chih-Neng Hsu, "Experimental and Performance Analyses of a Turbomolecular Pump Rotor System," *Vacuum*, vol. 121, pp. 260-273, 2015. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [14] Saddam Hussien et al., "Design and Analysis of Vertical Axis Wind Turbine Rotors," *International Journal on Recent Technologies in Mechanical and Electrical Engineering*, vol. 2, no. 9, pp. 54-62, 2015. [[Google Scholar](#)] [[Publisher Link](#)]
- [15] Sa. Kasmaiee, Si. Kasmaiee, and A. Farshad, "Unsteady CFD Simulation of a Rotor Blade under Various Wind Conditions," *Scientific Reports*, vol. 14, no. 1, pp. 1-23, 2024. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [16] S. Mauro et al., "CFD Modeling of a Ducted Savonius Wind Turbine for the Evaluation of the Blockage Effects on Rotor Performance," *Renewable Energy*, vol. 141, pp. 28-39, 2019. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [17] M.H. Mohamed, A.M. Ali, and A.A. Hafiz, "CFD Analysis for H-Rotor Darrieus Turbine as a Low-Speed Wind Energy Converter," *Engineering Science and Technology, an International Journal*, vol. 18, no. 1, pp. 1-13, 2015. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [18] M.H. Mohamed, A. Dessoky, and Faris Alqurashi, "Blade Shape Effect on the Behavior of the H-Rotor Darrieus Wind Turbine: Performance Investigation and Force Analysis," *Energy*, vol. 179, pp. 1217-1234, 2019. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [19] Nik Ahmad Ridhwan Nik Mohd, and George Barakos, "Performance and Wake Analysis of Rotors in Axial Flight using Computational Fluid Dynamics," *Journal of Aerospace Technology and Management*, vol. 9, no. 2, pp. 184-193, 2017. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [20] David Phan, and Dino Kondyles, "Rotor Design and Analysis; A Technique Using Computational Fluid Dynamics (CFD) and Heat Transfer Analysis," *21st Annual Brake Colloquium & Exhibition*, Hollywood, Florid, 2003. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [21] Sonu Sharma, and Rajesh Kumar Sharma, "Performance Improvement of Savonius Rotor Using Multiple Quarter Blades – A CFD Investigation," *Energy Conversion and Management*, vol. 127, pp. 43-54, 2016. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]

- [22] Sonu Sharma, and Rajesh Kumar Sharma, “CFD Investigation to Quantify the Effect of Layered Multiple Miniature Blades on the Performance of Savonius Rotor,” *Energy Conversion and Management*, vol. 144, pp. 275-285, 2017. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [23] Wenbo Shi et al., “CFD Analysis of Contrarotating Open Rotor Aerodynamic Interactions,” *International Journal of Aerospace Engineering*, vol. 2018, no. 1, pp. 1-13, 2018. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [24] Giuseppe Bruni, Sepehr Maleki, and Senthil K. Krishnababu, “Deep Learning Modeling of Manufacturing and Build Variations on Multistage Axial Compressors Aerodynamics,” *Data-Centric Engineering*, vol. 6, pp. 1-19, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [25] Matheus Costa Pereira, Mirelli de Castro Cesário, and Anderson Paulo de Paiva, “Dataset for Analysis of Computational Fluid Dynamics Results from an Industrial Centrifugal Fan,” *Data in Brief*, vol. 63, pp. 1-13, 2025. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [26] Tamás Orosz et al., “Automatic Tolerance Analysis of Permanent Magnet Machines with Encapsulated FEM Models Using Digital-Twin-Distiller,” *Processes*, vol. 9, no. 11, pp. 1-15, 2021. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]
- [27] Sayantan Sengupta, and Abhijit Guha, “Inflow-Rotor Interaction in Tesla Disc Turbines: Effects of Discrete Inflows, Finite Disc Thickness, and Radial Clearance on the Fluid Dynamics and Performance of the Turbine,” *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy*, vol. 232, no. 8, pp. 971-991, 2018. [[CrossRef](#)] [[Google Scholar](#)] [[Publisher Link](#)]