

Research Article

Bionic Riblet Impeller for a Centrifugal Pump: CFD and Experimental Performance Analysis

R. J. Pawar*

M. P. Ray

S. R. Suryawanshi

Department of Mechanical
Engineering, MET's Institute of
Engineering, Bhujbal Knowledge
City, Nashik, Maharashtra, 422003,
India

Received 26 June 2025

Revised 22 October 2025

Accepted 2 November 2025

Abstract:

This study adopts a novel bionic approach to evaluate the performance of a centrifugal pump using numerical simulations and validates the findings through experimental analysis. The centrifugal pump, featuring six backward-curved blades, was modeled by incorporating shark-skin-inspired riblet structures into the impeller design. The flow inside the pump was studied using k -epsilon (k - ϵ) turbulence models. The k - ϵ model gave results that were closest to the real experimental data. In most studies, only design parameters considered during computational fluid dynamics (CFD) analyses without consideration of experimental result. Here new bio inspired technique called bionics is applied to impeller of pump. The experiments were carried out using a computer-controlled pump testing setup. The pump's head, hydraulic efficiency, and power were analyzed using both CFD simulations and experimental tests on the duty point as per manufacturer's catalog, having the revolution of 2,900 rpm, The bionic impeller achieves a best efficiency point (BEP) at a discharge rate of 12 m³/hr., with a head of 32.75 m and an efficiency of 39.21%. The CFD results closely matched the experimental results at the design flow rate and at all other tested flow rates as per duty points. In addition, by integrating nature-inspired design with testing, this study contributes to the development of high-performance, sustainable pumps. The results may help advance efficient industrial fluid transport systems. This research bridges biomimetic innovation with practical engineering, paving the way for future pump advancements.

Keywords: centrifugal pump, biomimetic riblet structure, shark skin, CFD

1. Introduction

Centrifugal pumps are widely used for water supply, industrial use, wastewater handling, water treatment, irrigation, and other purposes [1]. A slight improvement in pump efficiency is possible to reduce the current 20% power consumption relationship of pumps worldwide [2]. There are two major issues in the world: lack of energy and deterioration of the environment. These issues make it more important than ever to improve the energy efficiency of the industry [3]. An area where a major improvement can be achieved is the operation of the centrifugal pump. These pumps are commonly used in hydroelectric power plants and in many industrial processes. They play an important role in liquid transfer, but also use a lot of energy. This means that even a slight improvement in efficiency can save a significant amount of energy. Centrifugal pumps are popular because they are cheap and flexible enough to be used. However, most current designs focus solely on hydraulic performance (how well the pump moves the liquid) or acoustic performance (the amount of noise in the pump) [4]. These two factors are usually treated as separate concerns, with little effort to optimize them together. This leaves room for improvement.

* Corresponding author: R. J. Pawar
E-mail address: pawarraj7938@gmail.com



An exciting field of innovation is biomimetic design systems inspired by nature. Scientists in particular have looked into how shark skin reduces air resistance in the water. Shark skin has a small ribbed structure called riblet, which helps sharks move the water more easily by reducing resistance [5]. This idea is already being used in other regions such as aircraft and ships to improve efficiency. However, only a limited range of studies have been studied how riblets, which inspired the skin of this house, can improve both the hydraulic and acoustic performance of centrifugal pumps [6]. Among tested nonsmooth vane shapes, the circular pattern showed the best antiwear effect, lowering vane wear while only slightly reducing head and efficiency [7].

This study focuses on the use of sharkskin-inspired riblet structures in centrifugal pumps. By adding these riblet structures to the impeller, the goal is to see how they affect important performance measurements such as pump head (liquid height), power consumption, and overall efficiency [8]. The study aims to fill a critical gap by not only improving how well the pump moves fluids but also addressing how quiet and efficient it can be at the same time. Previous studies have shown [9]. Microstructures with negative crown riblets can effectively reduce drag and flow resistance by stabilizing turbulent currents, whereas positive crowns intensify turbulence; therefore, the depth and placement of negative crowns are critical for optimizing drag-reduction performance [10]. Circular micro-grooves on vane surfaces reduced wear by altering particle trajectories, with little impact on hydraulic performance [11]. Shallow dimples on the suction-side front of blades improved efficiency and reduced drag by stabilizing boundary layers [12].

CFD provides inexpensive and accurate variations in scaling model testing, which allows for rapid simulations to be performed [13] [14]. The development of CFDs provides a detailed analysis of the flow structure of the pump [15]. In modern technology implementation, the design and development of mechanical components are increasingly shifting towards virtual platforms. Modern software tools such as CATIA, Autodesk Inventor, Solid Edge, SolidWorks, Siemens NX, and CFTurbo are now commonly employed to facilitate this transition. [16]. Simulation tools such as ANSYS, AutoCAD, FlexSim, CFD, and others are widely utilized to analyze fluid flow behavior. These tools help validate the performance and innovative aspects of mechanical components. [17]. Simulation tools offer detailed insights into the internal flow behavior of pumps, eliminating the immediate need for physical testing on actual equipment. They also allow analysis of various boundary conditions influencing pump performance. For instance, using ANSYS, by changing the number of impeller blades and the angle at which they exit, one can study how these changes affect the pump's efficiency and the pressure it produces [18]. Numerous investigations have concentrated on analyzing the flow patterns and pressure profile within the pump system, especially in impellers and spiral housing areas. However, numerical models' reliability confirmation, i.e., experimental review remains problematic [19]. Numerical models of pumps in the ANSYS program were compared with experimental measurements in wheel innovation [20]. In his study, numerical models were checked using available experimental data, the model's results can be effectively compared with real-world observations, often showing strong correlation. Parameters such as pressure and flow rate of liquids can be reliably measured using devices like pressure gauges and flow meters [21].

Centrifugal pumps are critical for industrial water transport, yet those operating at 10 m³/h and 2900 rpm, indicating inefficiencies in impeller design and power usage. This study explores the impact of bionic riblet structures on pump performance. Through experimental testing and numerical simulation, a comparison between conventional and bionic grooved impellers reveals that the bionic design enhances efficiency, reduces power consumption, and improves head, offering a promising approach for energy-efficient pump development.

2. Methodology and numerical simulation approaches

2.1 Bionic impeller

The subject of this study is a centrifugal pump operating at a constant rotational speed of $n = 2900$ rpm. The pump's design flow rate at its operating point is $Q = 10$ m³/h, with the primary geometric and hydraulic parameters of the impeller detailed in Table 1

Although various bionic groove designs have been explored, research on their effects in centrifugal pumps remains limited. Therefore, this study investigates the influence of shark skin-inspired bionic grooves on the hydraulic performance of centrifugal pumps. Bionic Groove Geometry (Fig. 1.) is used here

Table 1: Main Geometric Parameters

Components	Parameters	Symbols	Value	Unit
Impeller	Inlet diameter/mm	D_1	40	mm
	Outlet diameter/mm	D_2	25	mm
	Impeller diameter/mm	D	175	mm
	Blade number	z	6	

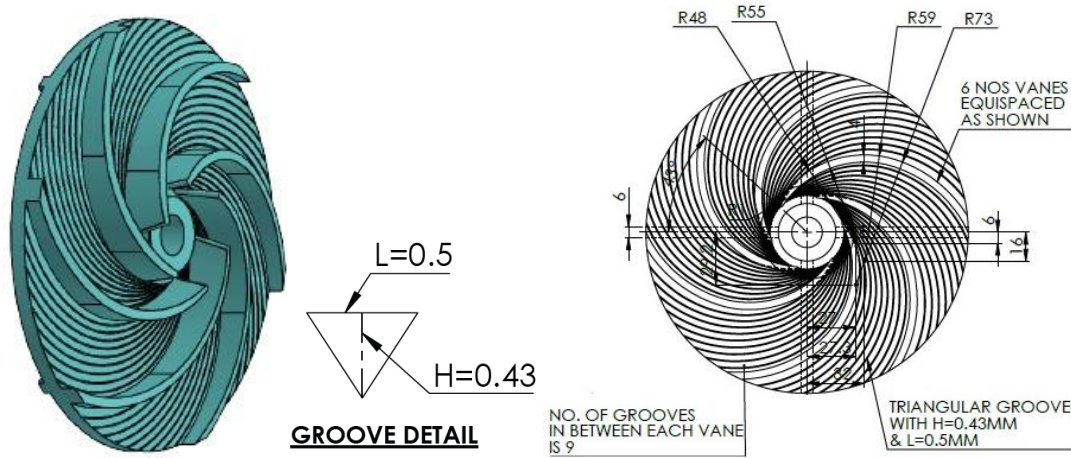


Fig. 1. Bionic groove geometry

2.2 Computational grid design

Grid generation, or meshing, is the process of dividing the flow domain into smaller control volumes. Most studies [22] use unstructured tetrahedral elements for meshing centrifugal pumps. Some researchers [23], [24] combine structured and unstructured elements, while others [25] use a mix of unstructured hexahedral and tetrahedral elements. Some applied tetrahedral grids for the double-volute casing and hexahedral grids for the impeller. In this study, all flow domains were meshed using unstructured tetrahedral grids, which are better suited for handling the complex geometry of the impeller and volute casing [26].

Fig.2. shows the grid of the model which is generated in ICEM CFD software. All domains are generated with unstructured tetrahedral grids. To capture the flow physics more accurately, the impeller domain was refined with a greater number of elements, which improves the reliability of the simulation results. The total number of elements created for the whole model is approximately 2.5 million.

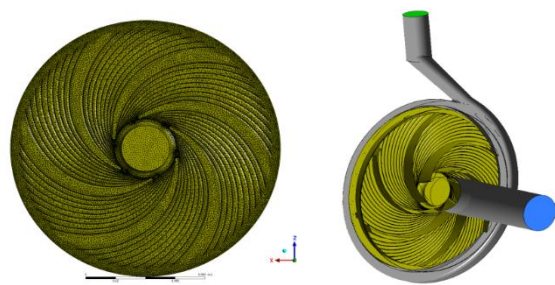


Fig. 2. Discretised computational domain

A mesh independence study was carried out with 1.8, 2.5, and 3.4 million elements, refined in the impeller domain. Variations in pressure head, efficiency, and torque between the medium mesh (2.5 million) and fine mesh (3.4 million) were within 1–1.5%, confirming mesh independence. Therefore, the medium mesh was adopted as it provided sufficient accuracy while maintaining computational efficiency, and with residuals below 1.0×10^{-5} (pressure residuals 10×10^{-6}), the results are both mesh-independent and numerically robust.

2.3 Numerical simulation

The Navier–Stokes equations provide a general framework for describing fluid flow. When applied to pumps and other flow machines, the basic Navier–Stokes equations must be adapted to represent the additional complexities introduced by rotation. These include momentum transfer and energy losses in rotating channels, which require modified turbulence models for accurate simulation. To model the flow in pumps more accurately, different turbulence models have been studied, including the SST $k-\omega$, and standard $k-\epsilon$. Among these, the $k-\epsilon$ turbulence model was determined to be the most suitable for performing Computational Fluid Dynamics (CFD) simulations of the centrifugal pump under investigation. The centrifugal pump operates at a high Reynolds number with predominantly turbulent internal flow. The standard $k-\epsilon$ model was chosen for its proven robustness and computational efficiency in predicting steady-state pump performance, providing reliable results for head, efficiency, and pressure distribution, as validated in previous studies [27],[28].

The impeller domain is modeled as a rotating fluid with water, a rotational velocity of 2900 RPM, isothermal conditions at 25°C, and a $k-\epsilon$ turbulence model. The casing domain is stationary with similar properties and turbulence model. Boundary conditions for the impeller include mass and momentum transfer interfaces, no-slip walls, and smooth wall roughness. The casing inlet has a total pressure of 1 bar, and the outlet specifies a mass flow rate, with no-slip walls on the casing surfaces. This setup ensures an accurate simulation of the centrifugal pump. The boundary conditions used in the analysis were based on real operational conditions, as outlined in Table 2.

Table 2: Boundary conditions for the present study

Position	Boundary Condition
Inlet	Mass flow inlet
Outlet	Pressure outlet
Impeller	Rotating reference frame
Interface	General Connection
Frame	Frozen Rotor
Wall at Casing	No slip Wall

The ANSYS FLUENT software was used to solve flow equations. The transport equations provided below are used to determine the turbulence kinetic energy and its dissipation rate.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_i} \left[\left(\mu + \frac{u_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (1)$$

and

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{u_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + C_{1\epsilon} \frac{\epsilon}{k} (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \frac{\epsilon^2}{k} + S_\epsilon \quad (2)$$

In these equations, G_k represents the production of turbulence kinetic energy resulting from mean velocity gradients, calculated according to the approach outlined in the $k-\epsilon$ Models. G_b is the generation of turbulence kinetic energy due to buoyancy, calculated as described in Effects of Buoyancy on Turbulence in the $k-\epsilon$ Models.

Y_M This term accounts for the influence of fluctuating dilatation in compressible turbulence on the overall dissipation rate, evaluated as per the methodology discussed in the $k-\epsilon$ Models treatment of compressibility effects. $C_{1\epsilon}$, $C_{2\epsilon}$ and $C_{3\epsilon}$ are constants. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ and, respectively. S_k and S_ϵ are user-defined source terms.

Turbulent Viscosity Representation

The turbulent (or eddy) viscosity is calculated by integrating the effects of various turbulence parameters as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\omega} \quad (3)$$

Where C_μ is a constant

The model constants. $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ , σ_k and σ_ε have the following default values

$$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_\mu = 0.09, \sigma_k = 1.0 \text{ and } \sigma_\varepsilon = 1.3$$

These default values are based on experimental data obtained from basic turbulent flow scenarios, such as common shear flows—including boundary layers, mixing layers, and jets—as well as from studies of decaying isotropic turbulence generated by grids. They have been found to work fairly well for a wide range of wall-bounded and free shear flows.

2.4 Experimental set up

Experimental set up shown schematically in Fig.3. The efficiency of a centrifugal pump is evaluated experimentally, focusing on how varying flow rates influence key performance parameters. The flow rate is controlled by adjusting the discharge valve, gradually reducing the effective diameter of the pump's discharge pipe from an initial size of 0.40 meters to near closure. Before initiating the pump operation, the elevation head is measured by determining the vertical distance between the reservoir water level and the discharge pipe centerline using a tape measure. During operation, elevation head variations are monitored using a vertical scale, with efforts made to maintain a consistent elevation head throughout the experiment.

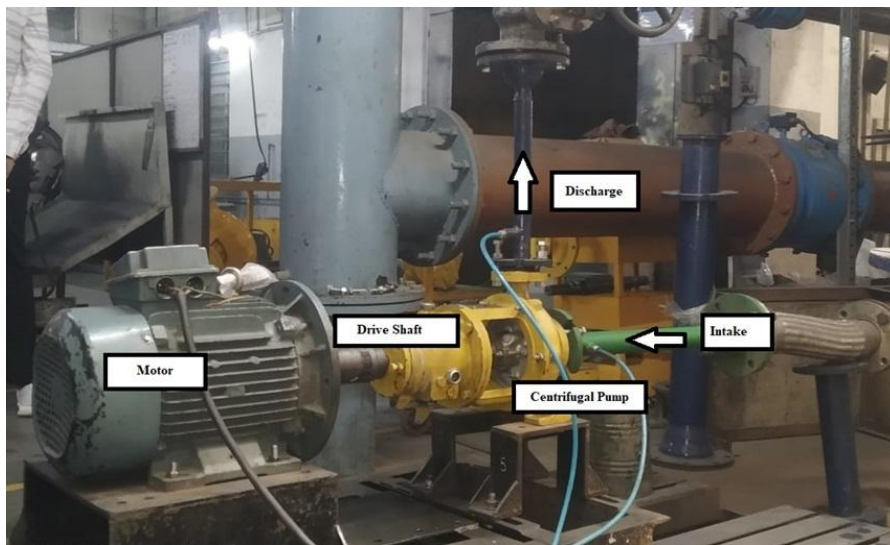


Fig. 3. Centrifugal pump test rig

To assess pump performance, key parameters such as input power, pressure head, and rotational speed are recorded. The pump is initially run until a steady-state condition is reached. A variable frequency drive (VFD) is used to measure the impeller's rotational speed in revolutions per minute (RPM). The pressure head is recorded using a pressure gauge installed on the discharge line, with readings taken in kg/cm^2 . The input power, representing the electrical energy supplied to the pump, is obtained from the test setup panel, which consists of a voltmeter and an ammeter, providing power consumption values in kilowatts.

Flow rate measurements are conducted using an electromagnetic flow meter, which operates based on the principle of electromagnetic induction. These meters offer highly accurate readings for water-based fluids due to the absence of moving parts, thereby minimizing mechanical wear and measurement errors. To vary the flow rate, the discharge valve is gradually adjusted while keeping the pump speed constant. Five specific operating conditions are selected based on the manufacturer's specifications and in accordance with IS: 13518-1992 standards.

The total head developed by the pump is determined using Bernoulli's principle, which considers the combined effect of elevation head, pressure head, and velocity head. In this study, head losses due to friction and flow disturbances caused by bends and fittings are assumed to be negligible for simplification. The Bernoulli equation, also known as

the equation of energy, is utilized to calculate the total head (H) generated by a pump. Bernoulli's equation relates the pressure, velocity, and elevation at two points along a streamline. It is given by equation

$$\frac{P_1}{\rho g} + \frac{V_1^2}{2g} + h_1 = \frac{P_2}{\rho g} + \frac{V_2^2}{2g} + h_2 + h_f \quad (4)$$

Where: P = Pressure at a point, N/m²
 ρ = Density of the fluid, kg/m³
 g = Acceleration due to gravity (9.81 m/s²)
 V = Velocity of fluid, m/s
 h = Elevation or height above reference point, m
 h_f = Head loss due to friction, m

Pump efficiency is evaluated by measuring the volumetric flow rate in six distinct trials, with each trial consisting of two replicates. The average of these replicates is considered for analysis. The output power imparted to the fluid by the pump is calculated using the product of flow rate, fluid density, gravitational acceleration, and total head. The actual power supplied to the pump is obtained directly from the test setup panel. Efficiency is then determined as the ratio of output power to input power, expressed as a percentage.

Considering negligible head loss (and correction factor, $\alpha=0.13$ m, head of the pump is given by equation (5) [29-31] as:

$$H = \frac{\Delta P}{\rho g} + \frac{V^2}{2g} + \Delta z \quad (5)$$

Where: H = Total head
 ΔP = Differential pressure, kg/cm²
 $\frac{\Delta P}{\rho g}$ = Pressure head, m
 $\frac{V^2}{2g}$ = Velocity head, m
 ρ = Water density (1000 kg/m³)
 Δz = Elevation head, m
 g = Gravitational acceleration (9.81 m/s²)

The pump power to the water (output power) P_{output} , is computed as by equation (6) [24,25].

$$P_{output} = \rho g Q H \quad (6)$$

Where: ρg = specific weight of water (9.81 kN/m³)

The input power P_{input} supplied to the pump is recorded from the pump test rig control panel. The efficiency of pump is calculated as the ratio of output power to the input power as in the equation below [23-25].

$$\eta = \frac{P_{output}}{P_{input}} \times 100 \quad (7)$$

Where: η = pump efficiency in percentage

3. Results and Discussion

In this study, the performance of a centrifugal pump was investigated through both experimental testing and computational fluid dynamics (CFD) simulations. The operating conditions were selected based on manufacturer specifications, with a constant rotational speed of 2900 rpm. The flow rate was varied by adjusting the discharge valve while maintaining a steady pump speed. Five specific operational conditions were considered for the analysis.

For the CFD simulations, the pump's rotational speed remained fixed while the volume flow rate was systematically varied from 0 to 12 m³/h. After achieving convergence, key parameters such as, turbulence kinetic energy, mass flow rate, turbulence dissipation rate, and velocity profiles were evaluated to ensure that the solution remained within an acceptable convergence criterion of 10⁻⁴.

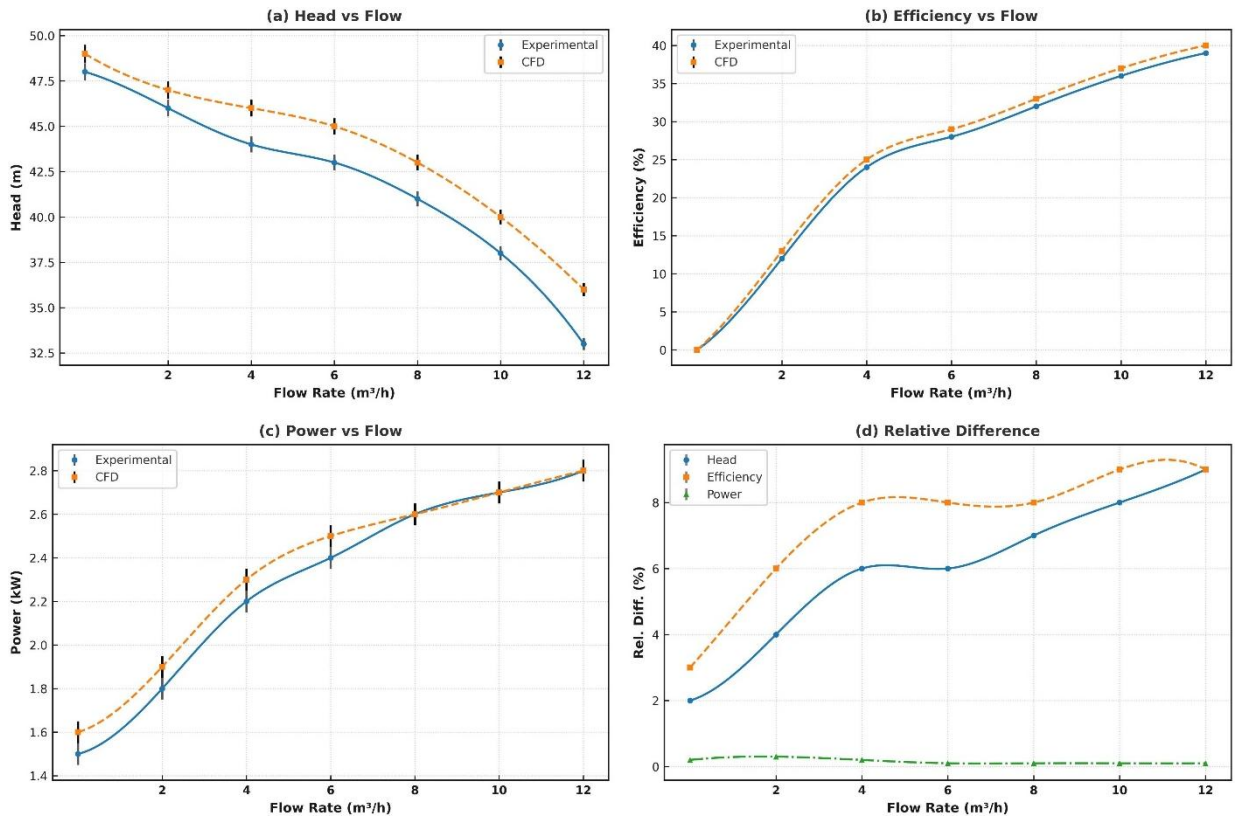


Fig. 4. Comparison between simulation and experimental results

The flow analysis was conducted using the k- ϵ turbulence model, which was applied at the design flow rate. When compared with experimental data, the k- ϵ turbulence model provided the most accurate results, demonstrating its effectiveness in predicting the pump's performance.

The simulation used a fluid-fluid interface, steady-state analysis, and a turbulence intensity of 5%, with water as the working fluid. The CFD and experimental findings for pump head and hydraulic efficiency at the design flow rate are illustrated in the figure. A detailed comparison of hydraulic efficiency and head with respect to volumetric flow rate, based on both experimental and numerical data, is depicted in Figures 4(a) and 4(b).

Figure 4(a) compares the head-flow performance curves of the bionic impeller pump obtained from experiments and CFD simulations. As is characteristic of centrifugal pumps, the head is highest at shut-off and declines progressively with increasing discharge. The CFD-predicted head curve aligns closely with the experimental results, demonstrating that the model captures the overall performance trend effectively. At the design flow, the agreement is especially strong, with discrepancies of only 1–2% at the best efficiency point (BEP). Minor deviations appear at the off-design extremes: at very low flow and especially toward the maximum flow (~12 m³/h), the CFD tends to slightly over-predict the head compared to experiment

Fig. 4(b) compares the pump's hydraulic efficiency obtained from experiments and CFD efficiency is low at zero flow, increases with flow, and reaches a peak before eventually declining at the highest flow rates. The peak of this curve identifies the best efficiency point (BEP). In this case, the BEP occurs at approximately $Q \approx 12$ m³/h, where the pump achieves its maximum efficiency. At this optimum, the experimental efficiency reaches about 39%, while the CFD predicts a slightly higher peak around ~42%. The CFD and experimental efficiency values at the BEP are

almost coincident) demonstrating that the simulation accurately pinpoints the optimal operating condition. Overall, the efficiency comparison shows a strong agreement in curve shape and peak location, with only small discrepancies at off-design points.

Fig. 4(c) illustrates the pump’s hydraulic output power as a function of flow rate. Fig. 4(d) illustrates the relative discrepancies between CFD simulations and experimental measurements for head, efficiency, and power across varying flow rates. Near the Best Efficiency Point (BEP), deviations are minimal less than 1% which validates the CFD model’s accuracy at the design condition. When the operating point shifts away from the BEP, discrepancies increase, though head differences remain limited to 5–7%. Overall, the CFD results demonstrate reliable accuracy, with all errors remaining below 9%.

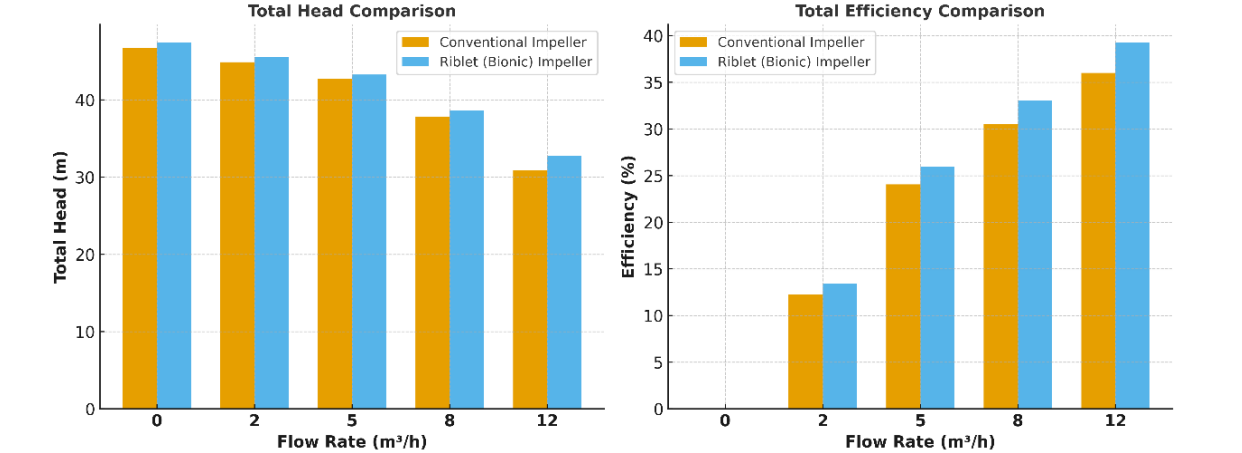
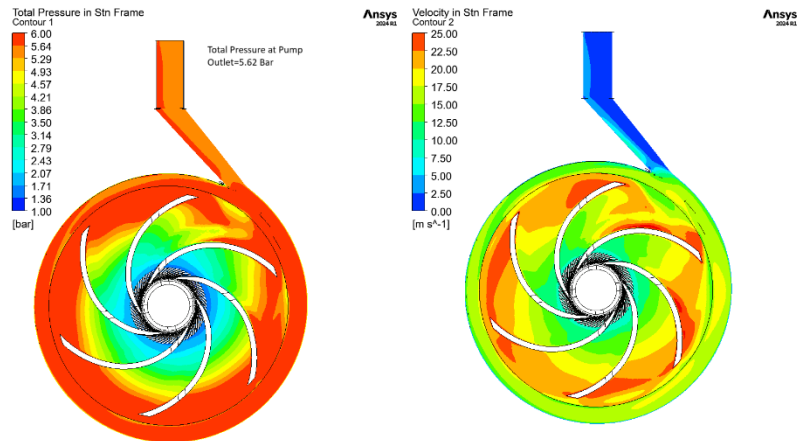


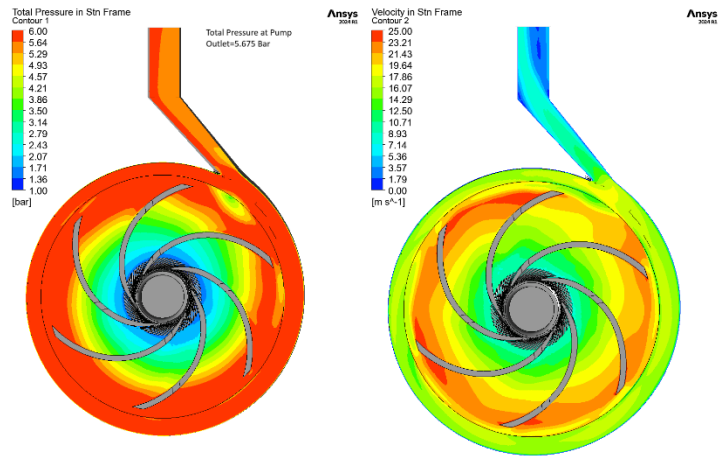
Fig. 5. Comparison of total head and efficiency between conventional and bionic impellers

The riblet impeller shows a clear improvement in both head and efficiency over the conventional design. The total head increased by 1.2–6.0 %, and the efficiency improved by 8–9 %, with the maximum efficiency rising from 35.99 % to 39.21 %. The BEP remained unchanged at 12 m³/h, indicating that the riblet modification enhanced energy conversion without affecting the operating stability. Overall, the riblet surface structure effectively reduced hydraulic losses, resulting in improved flow guidance and higher overall pump performance.

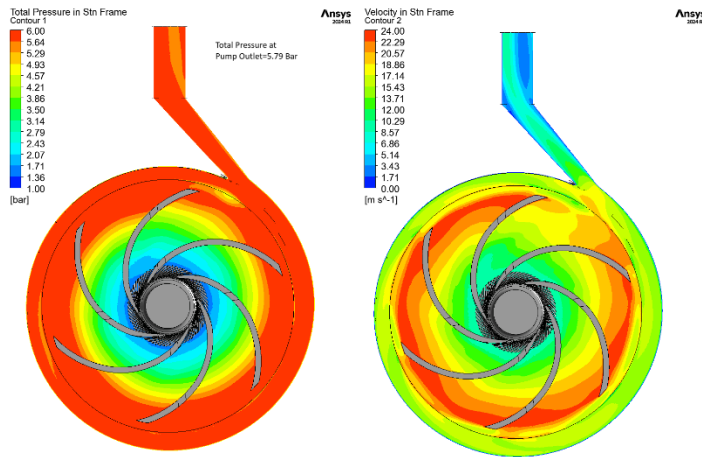


(a) Total pressure contours and velocity contours at a flow rate of 2 m³/h.

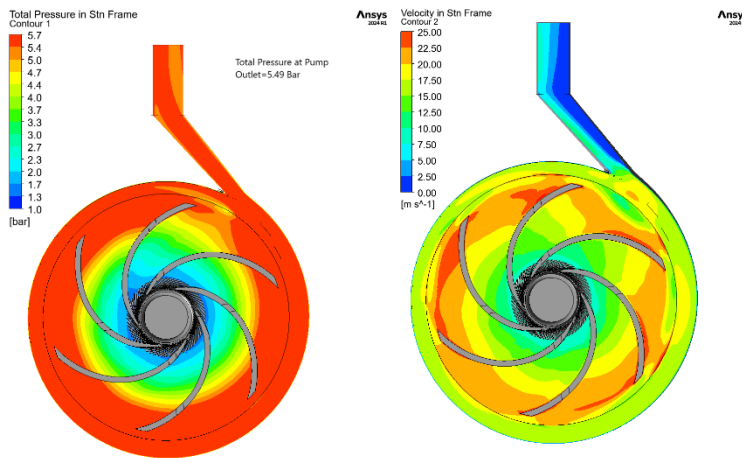
Fig. 6. Total pressure contours and velocity contours on XZ plane on flow rates of:
(a) 2 m³/h, (b) 5m³/h and (c) 8 m³/h (d) 10 m³/h (e) 12 m³/h



(b) Total pressure contours and velocity contours at a flow rate of 5 m³/h.

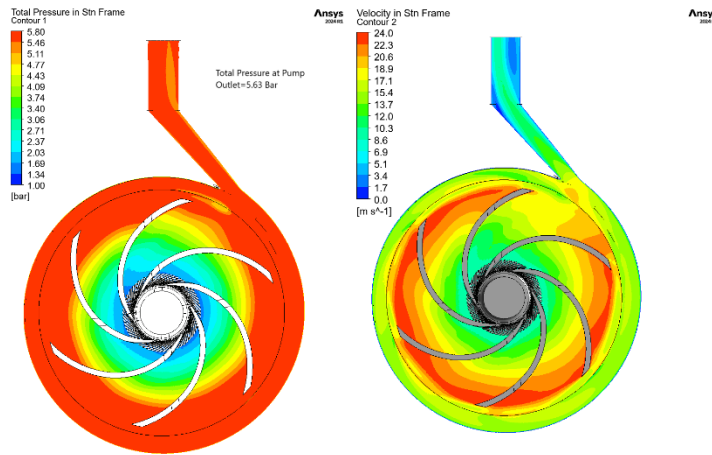


(c) Total pressure contours and velocity contours at a flow rate of 8 m³/h.



(d) Total pressure contours and velocity contours at a flow rate of 10 m³/h.

Fig. 6. Total pressure contours and velocity contours on XZ plane on flow rates of:
(a) 2 m³/h, (b) 5m³/h and (c) 8 m³/h (d) 10 m³/h (e) 12 m³/h



(e) Total pressure contours and velocity contours at a flow rate of 12 m³/h.

Fig. 6. Total pressure contours and velocity contours on XZ plane on flow rates of:
(a) 2 m³/h, (b) 5m³/h and (c) 8 m³/h (d) 10 m³/h (e) 12 m³/h

Two-dimensional total pressure contours and corresponding velocity contours in XZ cross-section plane are indicated in Fig. 6, at low flow conditions, the bionic impeller maintained a stable pressure field, characterized by smooth green/blue gradients, indicating low energy dissipation. The velocity plot showed minimal spikes and steady flow, suggesting reduced turbulence and more efficient energy usage as shown in Fig. 6(a).

As indicated in Fig. 6(b) with mid-range flow, the bionic impeller continued to minimize turbulence, showing well-distributed pressure zones and a uniform velocity field. This resulted in better energy conservation and less mechanical wear on internal components. At higher flow conditions, the bionic impeller maintained smoother pressure transitions, reducing high-gradient zones that typically indicate energy loss. Velocity vectors remained well directed and steady, limiting resistance and improving fluid dynamics as presented in Fig.6(c).

With increasing load, the bionic impeller continues to display stable pressure distributions and evenly spread velocity fields. Such characteristics limit hydraulic losses, reduce component stress, and enhance long-term operational reliability. This enhances operational reliability and reduces hydraulic losses in high flow environment as indicated in Fig. 6(d). It can be seen in Fig. 6(e) that as extreme duty, the bionic impeller continued to exhibit superior pressure handling with stable velocity fields, even under high stress. The bionic impeller sustains performance under extreme flow, reducing cavitation risk and energy loss, ensuring durable and efficient operation.

4. Conclusion

This study presents a comprehensive analysis of CFD simulations and experimental investigations conducted on a centrifugal pump equipped with a bionic impeller. The computational results were validated against experimental data obtained from a physical prototype manufactured by Indo Pump Pvt. Ltd, India. The key enhancements and findings from this research are summarized below:

- The integration of sharkskin-inspired riblet structures into the impeller design significantly improved hydraulic performance, demonstrating higher efficiency, increased head, and reduced energy losses compared to conventional designs.
- The computational fluid dynamics (CFD) analysis using the k-ε model closely matched experimental results across all duty points, particularly at the Best Efficiency Point (BEP), with error margins under 2% in critical parameters.
- The simulation setup, including boundary conditions and meshing strategy, proved effective in predicting real-world performance, making it a reliable tool for future impeller and pump design validation.

- The bionic impeller maintained uniform pressure and velocity distributions even under extreme flow conditions (up to 12 m³/h), thereby enhancing mechanical integrity and minimizing cavitation risks.
- The bionic impeller design achieved a peak hydraulic efficiency of 39.21% at the design flow rate, contributing to lower power consumption and promoting more sustainable fluid transport solutions in industrial applications.
- This study successfully demonstrates the practical application of bio mimicry in mechanical systems, bridging nature-inspired innovation with real-world engineering through both simulation and experimental validation.
- Future studies will focus on evaluating additional benefits such as noise reduction and cavitation suppression to further enhance pump performance.

Acknowledgements

This work is supported by MET's Institute of Engineering, Nashik, and Savitribai Phule Pune University, India, as well as Indo Pump India Limited, Ambad, Nashik.

Nomenclature

H	Total head, m
ΔP	Differential pressure, kg/cm ²
ρ	Density of fluid, kg/m ³
v	Velocity of fluid, m/s
g	Acceleration, m ³ /s
z	Elevation head, m
P _{output}	Output power, kW
P _{input}	input power, kW
η	Efficiency of pump
Q	Discharge, m ³ /s
P	Pressure at a point, N/m ²

References

- [1] Sahoo T, Guharoy A. Energy cost savings with centrifugal pumps. *World Pumps*. 2009;2009:35-37.
- [2] Hieninger T, Goppelt F, Schmidt-Vollus R, Schlücker E. Energy-saving potential for centrifugal pump storage operation using optimized control schemes. *Energy Efficiency*. 2021;14(2):23.
- [3] Liu H, Cheng Z, Ge Z, Dong L, Dai C. Collaborative improvement of efficiency and noise of bionic vane centrifugal pump based on multi-objective optimization. *Advances in Mechanical Engineering*. 2021;13(2):1-15.
- [4] Dai C, Guo C, Chen Y, Dong L, Liu H. Analysis of the influence of different bionic structures on the noise reduction performance of the centrifugal pump. *Advances in Mechanical Engineering*. 2021;21(3):886-895.
- [5] Baeten SRR, Kočovski A, Jovanova J, Sakes A. Characterization of shark skin properties and biomimetic replication. *Bioinspiration & Biomimetics*. 2024;19(5):051002.
- [6] Dean B, Bhushan B. Shark-skin surfaces for fluid-drag reduction in turbulent flow: A review. *Philosophical Transactions of the Royal Society A: Mathematical, Physical and Engineering Sciences*. 2010;368(1929):4775-4806.
- [7] Ma G, Xia L, Mou M, Wu Y, Yan Z. Effect of bionic nonsmooth surface vane on the antiwear characteristics of double-vane pump. *Applied Bionics and Biomechanics*. 2022;2022:1-13.
- [8] Wang Y, Dong L, Zhou R, Guo C, Dai C. Energy loss and noise reduction of centrifugal pump based on bionic V-groove geometry. *Water*. 2024;16(15):2183-2195.
- [9] Singh K, Singh A, Singh DK. Modelling and optimization of fluid frictional torque in a single-stage centrifugal pump with a vaned diffuser based on RSM, ANN, and desirability function. *Journal of Applied Fluid Mechanics*. 2025;18(3):728-741.
- [10] Ao M, Wang M, Zhu F. Investigation of the turbulent drag reduction mechanism of a kind of microstructure on riblet surface. *Micromachines*. 2021;12(1):59.

- [11] Ma L, Gu Y, Xia K, Mou C, Mou J, Wu D, Yan M. Influence of bionic circular groove blade surface on wear performance. *Lubricants*. 2022;10(5):101.
- [12] Liu J, Zhang F, Zhu LF, Yuan SQ, Xu RH, Zhang H. Influence of bionic structure on hydraulic performance and drag reduction effect of a centrifugal pump. *Journal of Physics: Conference Series*. 2024;2707(1):012044.
- [13] Siddique MH, Bellary SAI, Samad A. Design and optimization of pump impeller using response surface methodology and genetic algorithm. *Des Engineering*. 2017;2017(28):147-155.
- [14] Singh VR, Zinzuvadia MJ, Sheth SM. Parametric study and design optimization of centrifugal pump impeller: A review. *International Journal of Engineering Research and Applications*. 2014;4(1):216-220.
- [15] Yedidiah S. A study in the use of CFD in the design of centrifugal pumps. *Engineering Applications of Computational Fluid Mechanics*. 2008;2:331-343.
- [16] Chandrasekaran M, Santhanam V, Venkateshwaran N. Impeller design and CFD analysis of fluid flow in rotodynamic pumps. *Materials Today: Proceedings*. 2021;37:2153-2157.
- [17] Gülich JF. *Centrifugal pumps*. Berlin: Springer; 2010.
- [18] Sankar S. Analysis of centrifugal pump impeller using ANSYS. *International Journal of Innovative Research in Science, Engineering and Technology*. 2018;7:5021-5026.
- [19] Hassan A, Abdallah H, Abou El-Azm Aly A. Effect of impeller blade slot on centrifugal pump performance. *Global Journal of Research in Engineering*. 2016;16:71-85.
- [20] Alemi H, Nourbakhsh S, Raisee Dehkordi M, Najafi A. Effects of volute curvature on performance of a low specific-speed centrifugal pump at design and off-design conditions. *Journal of Turbomachinery*. 2015;137:041009.
- [21] Deshmukh D, Siddique MH, Samad A. Surface roughness effect on performance of an electric submersible pump. In: *ASME 2017 Gas Turbine India Conference*; 2017.
- [22] Siddique MH, Bellary SAI, Samad A, Kim JH, Choi YS. Experimental and numerical investigation of the performance of a centrifugal pump when pumping water and light crude oil. *Arabian Journal for Science and Engineering*. 2017;42(11):4605-4615.
- [23] Jafarzadeh B, Hajari A, Alishahi MM, Akbari MH. The flow simulation of a low-specific-speed high-speed centrifugal pump. *Applied Mathematical Modelling*. 2011;35(1):242-249.
- [24] Hedi ML, Hatema K, Ridha Z. Numerical flow simulation in a centrifugal pump. *International Renewable Energy Congress*. 2010;2010:300-304.
- [25] Kumar S, Mohapatra SK, Gandhi BK. Investigation on centrifugal slurry pump performance with variation of operating speed. *International Journal of Mechanical and Materials Engineering*. 2013;8(1):40-47.
- [26] Zhang H, Tang L, Zhao Y. Influence of blade profiles on plastic centrifugal pump performance. *Advances in Materials Science and Engineering*. 2020;2020:1-15.
- [27] Osman FK, Zhang J, Lai L, Kwarteng AA. Effects of turbulence models on flow characteristics of a vertical fire pump. *Journal of Applied Fluid Mechanics*. 2022;15(6):1661-1674.
- [28] Bai L, Zhang X, Zhang Y. Evaluation of various turbulence models and large eddy simulation for stall prediction in a centrifugal pump. *Water*. 2023;15(19):3432.
- [29] White FM. *Fluid mechanics*. 8th ed. New York: McGraw-Hill Education; 2015.
- [30] Karassik IJ, Messina JP, Cooper P, Heald CC. *Pump handbook*. 4th ed. New York: McGraw-Hill Education; 2008.
- [31] Fox RW, McDonald AT, Pritchard PJ. *Introduction to fluid mechanics*. 8th ed. Hoboken (NJ): John Wiley & Sons; 2011.