Volume 25 (1) 2026

Page 12 - 27

Mekanika: Majalah Ilmiah Mekanika

CFD Simulation Study on Airflow Dynamics Around a Cricket Ball: Effects of Velocity and Surface Modifications on Aerodynamic Performance

Catur Harsito[1], Enock Michael Kandimba[2], Raihan Danu Ramanda[3], Putra Adil Wicaksana[3], Yuki Trisnoaji[3], Singgih Dwi Prasetyo[3*]

1 Department of Mechanical, Computer, Industrial, and Management Engineering, Kangwon National University, Samcheok, South Korea
2 Department of Maritime Engineering, Dar es Salaam Ocean Institute, Dar es Salaam, Tanzania
3 Power Plant Engineering Technology, Faculty of Vocational Studies, State University of Malang, Malang, Indonesia

*Corresponding Author's email address: singgih.prasetyo.fv@um.ac.id

Keywords:
Cricket ball
CFD simulation
ANSYS Fluent
Aerodynamic forces
External airflow

Abstract
This study investigates the aerodynamic behavior of a cricket ball at various velocities using Computational Fluid Dynamics (CFD) in ANSYS Fluent, focusing on the effects of speed and surface modifications on aerodynamic forces and pressure distribution. The cricket ball geometry was simplified by replacing the seam with a protruding flat surface. Simulations were performed at airflow velocities of 20, 30, and 40 m/s using the realizable k-ε turbulence model, with air properties set to a density of 1.225 kg/m³ and dynamic viscosity of 1.81×10^{-5} Pa-s. At 20 m/s, the inlet and outlet mass flow rates were 50.306891 kg/s and -50.306901 kg/s, with a net imbalance of -9.3×10^{-6} kg/s, generating a drag force of 0.5 N, a lift force of 0.2 N, and a pressure difference of 50 Pa. At 30 m/s, the inlet and outlet rates were 75.460373 kg/s and -75.464958 kg/s, respectively, resulting in a net imbalance of -0.004585 kg/s. The flow was fully turbulent, producing a drag force of 3.5 N, a lift force of 1.5 N, and a pressure difference of 250 Pa. Increasing velocity boosts drag, lift, and pressure differences. At the same time, the flat surface enhances asymmetry, vortices, and swing at higher speeds.

1 Introduction

The dynamics of a cricket ball during delivery play a crucial role in determining the game's outcome. Key factors such as ball velocity, seam orientation, and environmental conditions influence its interaction with the surrounding airflow [1]. Understanding the aerodynamic behavior of a cricket ball is therefore essential for optimizing game strategies and informing ball design [2]. Phenomena such as swing and drift are direct consequences of the ball's aerodynamic properties, and these are strongly linked to the physics of airflow interaction with the ball's surface [3,4]. Numerous studies, both experimental and numerical,

https://dx.doi.org/10.20961/mekanika.v25i1.104182

Revised 17 September 2025; received in revised version 20 September 2025; Accepted 21 September 2025
Available Online 30 April 2026

2579-3144

have investigated the aerodynamics of cricket balls [5,6]. However, experimental investigations often involve substantial costs and lack flexibility in controlling parameters [7].

Computational Fluid Dynamics (CFD) provides an efficient alternative for studying the external flow field around the ball in greater detail [8]. Among CFD tools, ANSYS Fluent offers comprehensive capabilities for simulating a range of operating conditions, including velocity variations influenced by boundary conditions. In professional cricket, the speed of a ball varies considerably depending on the bowler's style and technique [9,10].

Fast bowling is typically performed at speeds of 135-160 km/h (37.5-44.4 m/s) and is achieved through a combination of body strength, momentum, and efficient transfer of kinetic energy during delivery. Medium-fast bowling ranges from 110-135 km/h (30.5-37.5 m/s), while slow bowling or spin bowling ranges from 70-95 km/h (19.4-26.4 m/s). The physical condition of the bowler, including muscle strength, flexibility, and coordination, further influences delivery speed [11,12]. These biomechanical and physiological factors play a significant role in determining the initial velocity imparted to the ball, which, in turn, affects its aerodynamic behavior during flight. Consequently, variations in delivery style and speed can produce markedly different aerodynamic responses in terms of drag, lift, and swing generation [13,14].

The aerodynamic forces acting on a cricket ball are primarily governed by drag and lift. Drag force, which represents the resistance of the airflow to the ball's motion, consists of pressure drag and skin friction drag [15,16]. The Magnus effect, induced by ball rotation, creates asymmetric airflow patterns that generate lift through pressure differentials, resulting in lateral deviation in the ball's trajectory. The surface condition of the ball can trigger a transition in the boundary layer from laminar to turbulent flow at specific velocities [17,18]. In addition, the seam plays a critical role in facilitating controlled turbulence, which enhances swing [19,20]. Understanding these aerodynamic mechanisms is crucial for accurately predicting the ball's flight path and developing effective bowling strategies. These principles also have implications for ball design, particularly in modifying surface features to achieve desired aerodynamic effects.

Although prior research has addressed the aerodynamics of various sports balls, including cricket balls, most studies have focused on experimental wind-tunnel measurements or numerical models of generic spherical or near-spherical geometries. There is a lack of studies specifically analyzing cricket ball aerodynamics that combine multi-speed CFD simulations with surface modification effects, particularly replacing the seam with a flat protrusion, to systematically examine pressure distribution, vortex formation, and the transition between laminar and turbulent flow. This gap is addressed in the present work through three-dimensional CFD simulations in ANSYS Fluent at multiple velocities (20, 30, and 40 m/s), which enable the quantification of drag and lift variations under different delivery conditions. The study is designed to provide a rigorous understanding of how flow dynamics around a cricket ball are influenced by both velocity and surface geometry. The results are expected to contribute to the advancement of aerodynamic ball design, the refinement of professional bowling techniques, and the development of performance optimization strategies in cricket [21,22].

Previous studies have examined the airflow dynamics around sports balls using both experimental and numerical approaches. Table 1 presents the methods, study objects, and main findings from each investigation, offering a comparative perspective on prior work in this field. The literature reviewed encompasses a range of ball types, surface configurations, and flow conditions, enabling the identification of key aerodynamic factors influencing ball performance. By synthesizing prior studies, the table provides context for the current research and highlights the methodological and thematic differences this study addresses. This structured comparison also reinforces the novelty of the present investigation, as it focuses on specific cricket ball conditions that have not been extensively analyzed in earlier research. Most previous research has focused on the aerodynamic analysis of sports balls such as soccer balls, baseballs, and tennis balls, emphasizing the influence of panel geometry, seam patterns, and surface texture on drag and lift forces. However, studies specifically investigating cricket ball aerodynamics using CFD, particularly those evaluating the combined effects of varying airflow velocities and surface modifications (such as replacing the seam with a flat protrusion) on pressure distribution, flow patterns, and vortex formation, remain limited.

Table 1. Previous research related to airflow dynamics on balls

Reference	Method	Object	Findings
[23]	3D modeling in SolidWorks 2019, CFD simulation in ANSYS Fluent 2019, validation with wind tunnel data from literature	Smooth sphere and 32-panel soccer ball	Surface geometry (panels & seams) influences the drag coefficient and flight path; panel size and attachment method significantly affect the ball's aerodynamic properties.
[24]	Numerical CFD simulation including gravity effects; validated with wind tunnel experiments	Soccer balls with varying panel numbers (32-panel, 14-panel, etc.) and seam types	Drag coefficient varies with panel number and surface texture; the 32-panel ball shows an earlier transition to a turbulent boundary layer, delaying separation; the drag crisis occurs between 12 and 15 m/s.
[25]	CFD (ANSYS Fluent) with standard *k-epsilon* model, transient solver, varying Reynolds numbers, and rotational conditions	Baseball and tennis balls	Surface roughness height delays boundary-layer separation; spin produces the Magnus effect; CFD simulations accurately predict trajectories and aerodynamic forces compared with experiments.
[26]	Large Eddy Simulation (LES) in ANSYS Fluent, modeling Adidas Telstar 18 (6 panels) and a smooth sphere	Adidas Telstar 18 soccer ball & smooth sphere	LES captures turbulent flow details and panel/seam effects; seams influence boundary layer separation points; panel design affects pressure distribution and ball trajectory.
[27]	CFD unsteady Reynolds-Averaged Navier-Stokes (RANS) with realizable *k-epsilon* model, rotating ball simulation in virtual wind tunnel	Baseball (four-seam fastball)	Lift force varies by up to 20% and drag force by up to 10% due to seam-orientation changes during rotation; seams affect boundary-layer separation and cyclically enhance/reduce the Magnus effect.

This study offers a novel contribution by integrating three-dimensional numerical simulations in ANSYS Fluent with multi-speed analysis (20, 30, and 40 m/s), providing an in-depth understanding of laminar-turbulent flow transition, pressure asymmetry induced by surface modifications, and their implications for lift generation and swing behavior in cricket balls. The approach not only enriches the existing body of sports aerodynamics literature but also serves as a strategic reference for designing more aerodynamic cricket balls and developing optimized bowling techniques in professional gameplay.

2 Methodology

This study employs a CFD-based numerical simulation method using ANSYS Fluent software to analyze the dynamics of airflow around a cricket ball [28,29]. This study aims to understand how various pitching conditions affect the airflow patterns and aerodynamic forces of the cricket ball, as illustrated in Figure 1.

The boundary conditions applied in the simulation are depicted in Figure 2, where the fluid domain is modeled with an inlet representing the constant air velocity specified by the test scenario, an outlet at atmospheric pressure, and walls enclosing the computational domain. The cricket ball is positioned at the center of the domain, with its surface defined as a stationary wall, while the airflow passes around it to replicate real-game conditions. This configuration ensures the accurate analysis of pressure distribution, vortex formation, and drag-lift behavior under various velocity conditions.

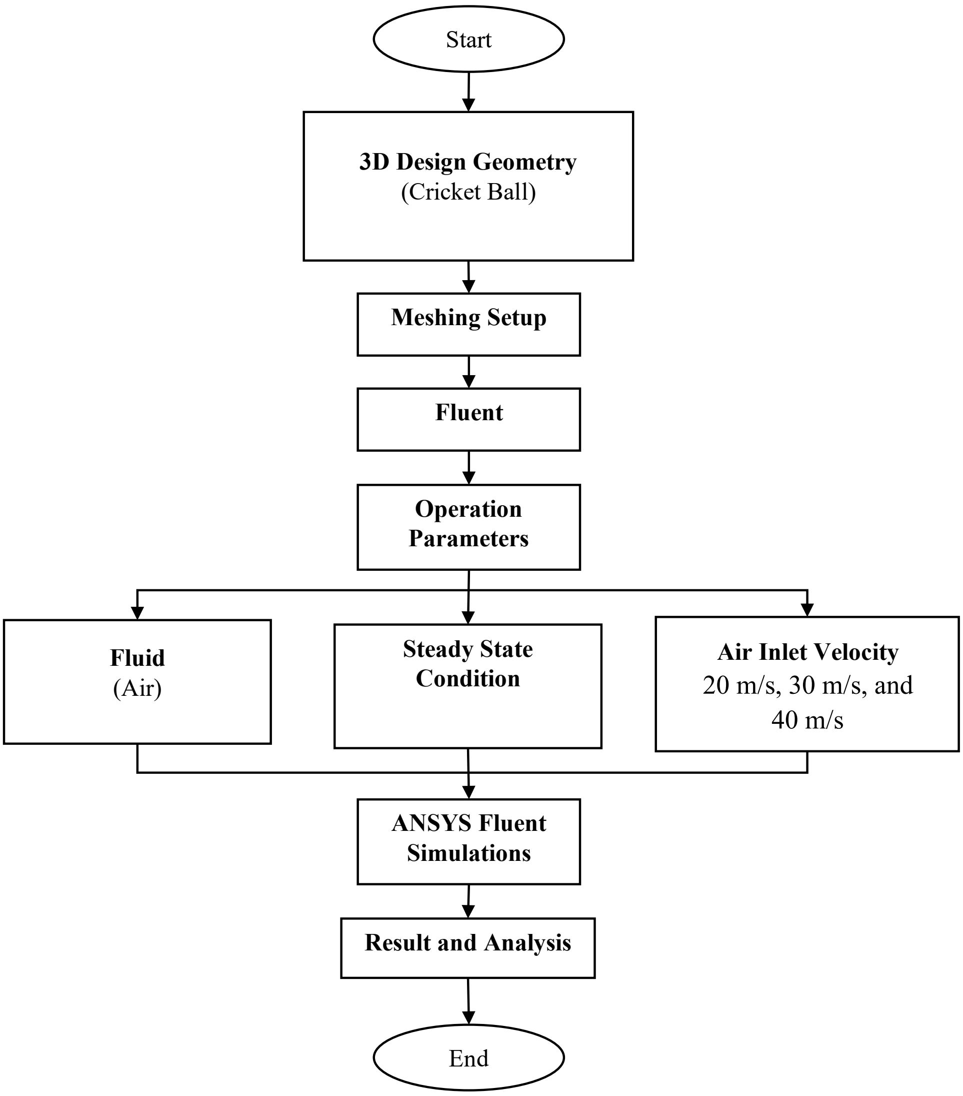


Figure 1. Research flow diagram

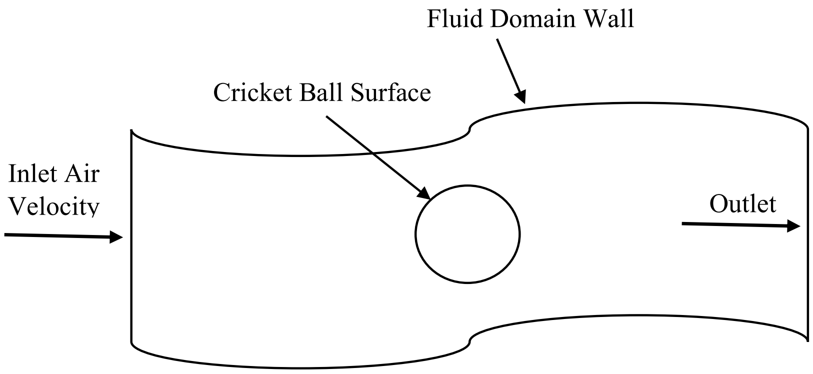


Figure 2. Boundary conditions for the current work

2.1 Mesh file, geometry design, and modeling

The cricket ball is modeled in three dimensions, accounting for the standard dimensions used in cricket matches. The cricket ball geometry model was simplified by replacing the original seam with a flat surface having a slightly higher diameter to streamline the analysis [30]. The model was taken from the ANSYS Innovation Course website, which provides the geometry and mesh references required for the simulation [31]. The seam orientation on the cricket ball was modified in the simulation to reflect the various pitching conditions that may occur during a match. This includes variations in seam orientation to simulate the effects of different bowling techniques, such as swing bowling and seam bowling [32,33].

2.2 Simulation setup in ANSYS Fluent

- Flow model: The k-ε Realizable turbulence model was used to predict the airflow pattern around a cricket ball at high speeds. This model was chosen because it can handle complex turbulent flows and provide realistic predictions of flow patterns.
- Boundary conditions:
  - Cricket ball: In this simulation, the cricket ball is considered stationary, while the fluid domain is a moving air duct with variable velocities (20, 30, and 40 m/s).
  - Inlet: Constant air velocity according to the scenario being tested [9].
  - Outlet: Atmospheric pressure is used as the boundary condition at the outlet to reflect the actual environmental conditions [34].
- Material properties: Air is considered a Newtonian fluid with standard physical properties, namely a density of 1.225 kg/m³ and a dynamic viscosity of 1.81×10^{-5} Pa-s [35]. The use of these standard physical properties aims to simplify the simulation and ensure comparable results with existing literature [36,37].

2.3 Data analysis

In the discussion section, you should explain and compare the results you obtained with the published paper or existing literature review. The findings and their implications should be discussed in the broadest context possible. Future research directions may also be highlighted. Pressure distribution: The CFD simulation will generate pressure distribution data around the cricket ball. This data will be analyzed to understand how pressure variations around the ball cause the lift and drag forces acting on it. Aerodynamic forces: Drag and lift are calculated from simulation data to quantify how airflow affects the cricket ball as it moves through the air. These forces are essential for determining the ball's performance under real game conditions.

4. Results and Discussion

4.1 Validation and sensitivity study

The simulation results will be validated by comparing them with existing literature data or similar experimental results from previous studies. This validation is crucial to ensure that the simulation model produces accurate predictions. Sensitivity Study: Sensitivity studies are conducted to ensure the stability of the simulation results by varying numerical parameters such as mesh size and convergence tolerance. It aims to determine how changes to the parameters affect the simulation results. This methodology is designed to provide accurate and in-depth simulation results regarding the external airflow dynamics around a cricket ball, particularly under real game conditions [38]. Using this approach, a better understanding of the factors affecting the performance of a cricket ball under various pitching conditions is expected [39,40].

Based on the validation results presented in Figure 3, the simulation data obtained in this study demonstrate an excellent agreement with findings from previous research. The comparison shows that the drag force from the reference was 1.45 N, while the simulation yielded 1.50 N, a difference of only 3.45%.

Similarly, the lift force value from the reference was 0.78 N, compared to 0.80 N from the simulation, with a difference of 2.56%. These relatively small discrepancies suggest that the CFD model employed in this study accurately represents the aerodynamic phenomena surrounding a cricket ball. Therefore, the model can be considered valid and reliable for further analysis of pressure distribution, flow patterns, and vortex formation under various velocity conditions. This agreement also confirms that the mesh configuration, boundary conditions, and turbulence model were implemented correctly, resulting in predictions consistent with both experimental results and previous simulations.

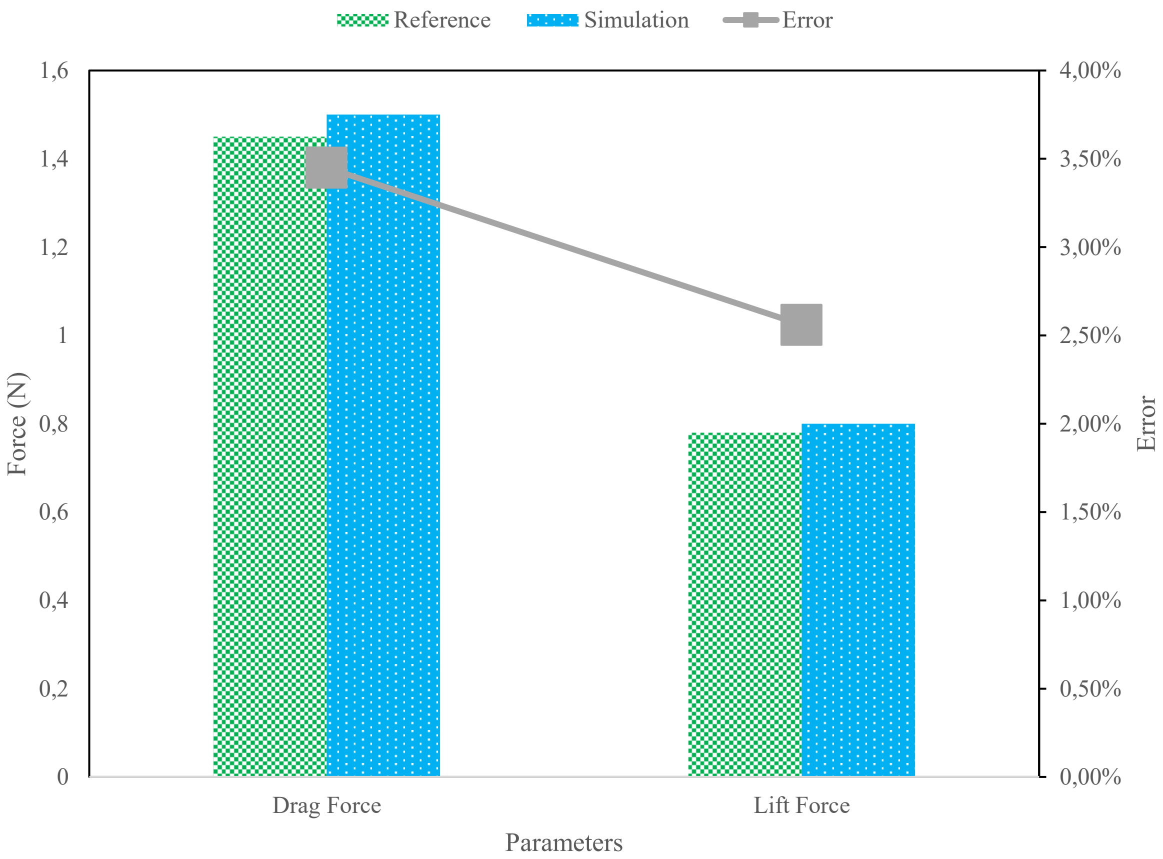


Figure 3. Validation with other research [41]

4.2 Result analysis

Based on the simulation of the external airflow around the cricket ball with three different speeds (20, 30, and 40 m/s), the following results and analysis for each speed condition are shown in Figures 4-6. Speed 20 m/s: At this speed, the pressure distribution around the ball shows a significant pressure difference between the front and back of the ball. This creates a considerable drag force, but a relatively low lift force. The airflow tends to be stable with minimal vortex patterns around the ball. Speed 30 m/s: With an increase in speed to 30 m/s, the pressure distribution shows an increase in drag and lift forces. The airflow pattern exhibits turbulent flow, with more complex vortices around the sphere. The pressure on the front of the ball increases, leading to a larger pressure difference and greater aerodynamic forces. Speed 40 m/s: The simulation's drag and lift forces reach their maximum values at this highest speed. The airflow around the ball becomes highly turbulent, characterized by larger and irregular vortices. The pressure difference between the ball's front and back sides becomes excessively high, creating strong aerodynamic forces that significantly affect the ball's direction and speed. These simulation results indicate that airflow velocity significantly affects the aerodynamic force on the cricket ball. Further analysis is required to understand the practical implications of these findings in the context of a real cricket game. For 20 m/s, the mass flow rate data in Table 2 and Figure 4 show an inlet value of 50.306891 kg/s and an outlet value of -50.306901 kg/s, resulting in a negligible net imbalance of -9.3178824 × 10^{6} kg/s.

Table 2. Mass flow rate of the velocity of 20 m/s

Parameters	Value (kg/s)
Inlet	50.306891
Outlet	-50.306901
Net	-9.3178824e-06

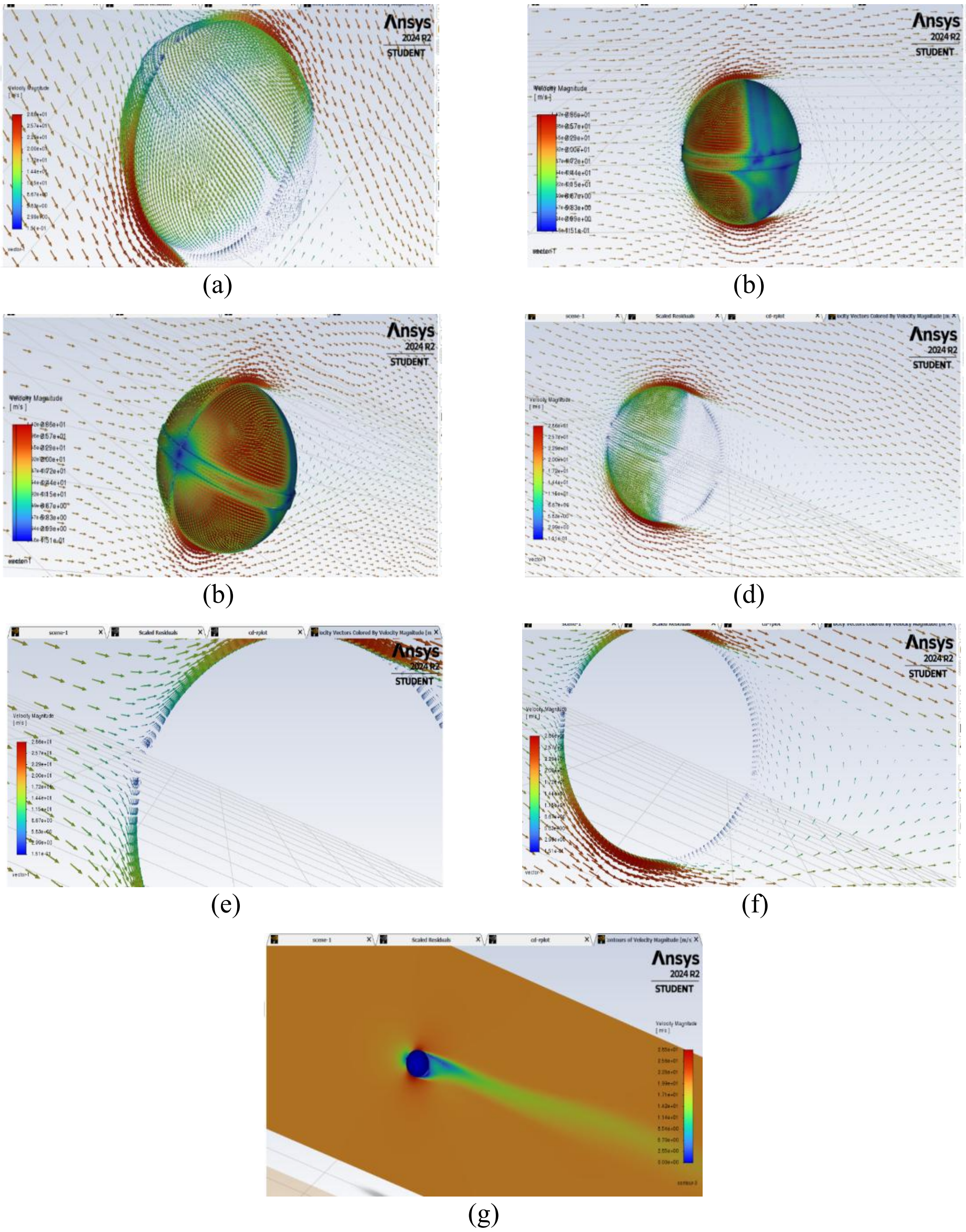


Figure 4. Simulation results in ANSYS Fluent with a velocity of 20 m/s: (a) Velocity contour of airflow around the ball, (b) Pressure on front and rear surfaces, (c) Velocity vector showing boundary layer and flow separation, (d) Streamline of wake formation, (e) Velocity near upper region, (f) Velocity near lower region, and (g) Static pressure and velocity field in the downstream wake

This near-zero difference confirms that the simulation satisfies mass conservation and that numerical errors due to discretization and convergence tolerance are minimal, thereby validating the reliability of the CFD setup for subsequent aerodynamic force calculations. The stability of the mass flow rate at this velocity aligns with the predominantly laminar flow and symmetric pressure distribution observed in the simulation, indicating that the mesh resolution and boundary conditions are adequate to capture steady-state airflow behavior around the ball. This agreement between numerical stability and physical flow characteristics reinforces the accuracy of the results for the 20 m/s case before extending the analysis to higher velocity conditions. At low speeds, the airflow around a cricket ball shows features that influence its aerodynamic behavior. The pressure distribution is nearly symmetrical, with the highest pressure at the stagnation point in front, where air first contacts the ball. At the rear, airflow separates and forms a wake, creating a low-pressure zone. The resulting pressure difference is slight, producing less drag and contributing to the ball's stable flight. The flow pattern is predominantly laminar, in which air layers move smoothly and in parallel, with little mixing or disturbance. In this state, aerodynamic resistance is minimal, and stability is higher. A flat surface (replacing the seam in a simplified model) introduces only slight disruption. At the back, separation still occurs, forming a small vortex, but not enough to cause swing. Swing requires asymmetric airflow and stronger vortices, which are unlikely at low speeds because laminar flow predominates.

Due to the dominance of laminar flow and the minimal transition to turbulence, the aerodynamic drag at low speeds is relatively low. Drag force is the sum of pressure drag and skin-friction drag. Pressure drag results from the pressure differential between the front and rear of the ball, while friction between the air and the ball's surface causes skin-friction drag. At low speeds, the pressure drag is minimal due to the slight pressure differential, and skin friction drag is also low because the smooth laminar flow reduces friction. Consequently, the ball experiences less air resistance as it travels, resulting in a more stable and predictable trajectory. This stability helps maintain a consistent path with minimal deviation, making it easier for bowlers to control their deliveries. Several key differences emerge when comparing airflow characteristics at low and high speeds. At higher speeds, the flow transitions from laminar to turbulent much earlier, leading to increased turbulence and larger vortices. This transition significantly affects the pressure distribution around the ball, resulting in a larger pressure differential between the front and rear. As a result, the aerodynamic drag and lift forces increase, making the ball's trajectory more complex and less predictable. In contrast, the low-speed environment allows for a more controlled and stable airflow, with less turbulence and fewer aerodynamic forces acting on the ball [42]. This stability is particularly beneficial for bowlers who rely on precision and consistency in their deliveries. The minimal aerodynamic drag at low speeds helps the ball maintain its intended path, making it easier to target specific areas on the pitch and execute strategic plays.

Understanding airflow behavior at low speeds has practical implications for both bowlers and cricket ball manufacturers. Bowlers can utilize the knowledge of low-speed aerodynamics to refine their techniques and improve their accuracy. By recognizing the conditions under which the ball exhibits stable flight, they can adjust their bowling strategies to exploit these characteristics. For instance, bowlers may focus on maintaining a consistent low-speed delivery to achieve greater control over the ball's movement. Manufacturers can also benefit from this understanding by designing cricket balls that enhance the desired aerodynamic properties at low speeds. Optimizing the ball's surface texture and material can influence the airflow pattern and reduce unwanted aerodynamic disturbances. This can result in cricket balls that offer better performance and consistency, particularly in conditions where low-speed deliveries are prevalent. In conclusion, the airflow around a cricket ball at low speeds is characterized by a symmetrical pressure distribution, laminar flow, and low aerodynamic drag. These features contribute to the stability and predictability of the ball's trajectory, providing advantages for bowlers who seek precision and control in their deliveries. The minimal pressure differential and reduced turbulence ensure the ball moves steadily through the air without significant deviations.

By leveraging this understanding, bowlers can enhance their techniques and manufacturers can design better-performing cricket balls. The insights gained from studying low-speed aerodynamics can be applied to various aspects of the game, ultimately improving cricket's overall performance and enjoyment. Whether through strategic bowling or innovative ball design, understanding airflow dynamics at low speeds is crucial to the sport. For the 30 m/s case, the mass flow rate results in Table 3 indicate an inlet value of 75.460373 kg/s and an outlet value of -75.464958 kg/s, producing a net imbalance of -0.0045852661 kg/s. Although the net difference is slightly larger than in the 20 m/s case, it remains within an acceptable tolerance range for CFD simulations and still satisfies the principle of mass conservation. The slight deviation can be attributed to increased turbulence intensity and more complex vortex structures observed at this speed, which introduce minor numerical fluctuations in the flow field. Nevertheless, the accuracy of the mass balance confirms that the mesh quality and boundary conditions remain appropriate for capturing the turbulent airflow regime at 30 m/s (see Figure 5), thereby supporting the validity of subsequent analyses of aerodynamic forces and pressure distributions at this velocity.

Table 3. Mass flow rate of the velocity of 30 m/s

Parameters	Value (kg/s)
Inlet	75.460373
Outlet	-75.464958
Net	-0.0045852661

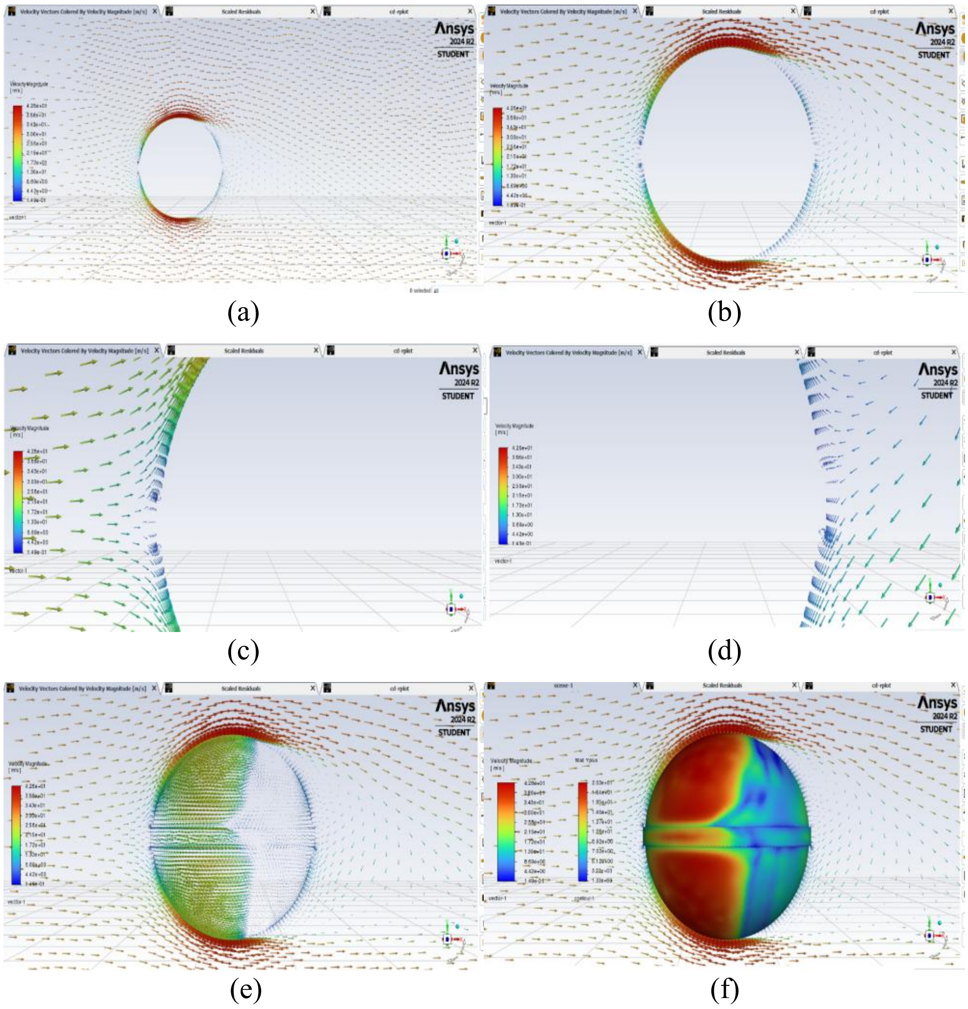
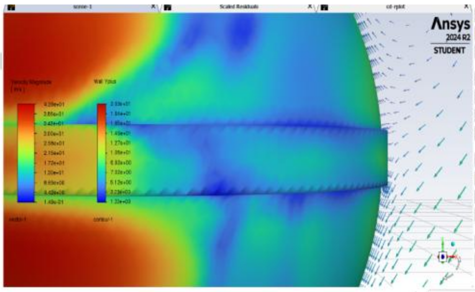
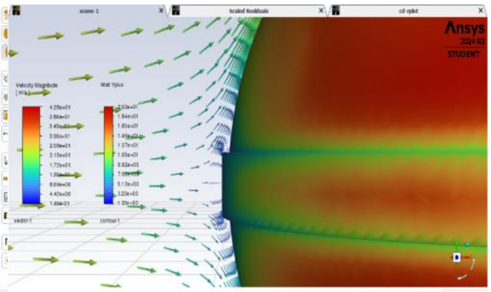
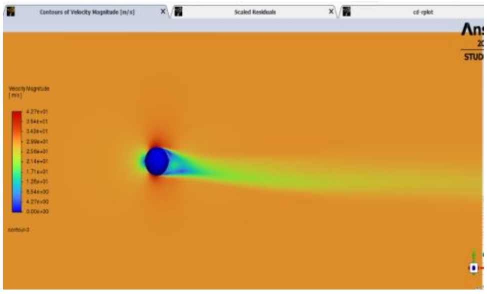


Figure 5. Simulation results in ANSYS Fluent with a velocity of 30 m/s: (a) Velocity vectors, (b) Pressure gradient front–rear, (c) Upper flow separation, (d) Lower delayed separation, (e) Asymmetric wake, (f) Laminar–turbulent transition, (g) Front–rear pressure contrast, (h) Flow deflection by seam, and (i) Low-pressure wake





(g) (h)



(i)

Figure 5. *Cont.*

When a cricket ball moves at medium speed, the surrounding airflow pattern becomes more complex than at low speed. Although high pressure remains in the area in front of the ball at medium speeds, the pressure difference between the front and back sides is greater than when the ball moves at a speed of 20 m/s. This is due to increased velocity, which results in a greater pressure force in front of the ball and a lower pressure force behind it, caused by the separated flow. The sphere's flat surface leads to the formation of larger vortices than at low velocities. The transition of flow from laminar to turbulent occurs on one side of the sphere, resulting in a more complex flow pattern. The resulting vortices begin to influence the ball's direction of movement, amplifying the drag forces acting on it. At this speed, lift forces become apparent due to a more pronounced pressure difference between the top and bottom of the ball.

This lift force can affect the ball's direction, especially if it is given spin on the throw. In addition, the force of drag increases significantly compared to low speeds, due to the more dominant turbulent flow around the sphere. At medium speeds, the swing effect starts to appear more clearly due to the interaction between the vortex and the pressure difference generated by the flat surface of the ball. The bowler can use this swing to fool the batter with a sudden change in ball direction. By understanding the dynamics of airflow at medium speeds, bowling strategies can be adjusted to maximize the swing effect and increase the chances of getting wickets [43]. For the 40 m/s case, the mass flow rate data in Table 4 and Figure 6 show an inlet value of 100.6138 kg/s and an outlet value of -100.61006 kg/s, resulting in a net imbalance of -0.0062561035 kg/s.

Table 4. Mass flow rate of the velocity of 40 m/s

Parameters	Value (kg/s)
inlet	100.6138
outlet	-100.61006
net	-0.0062561035

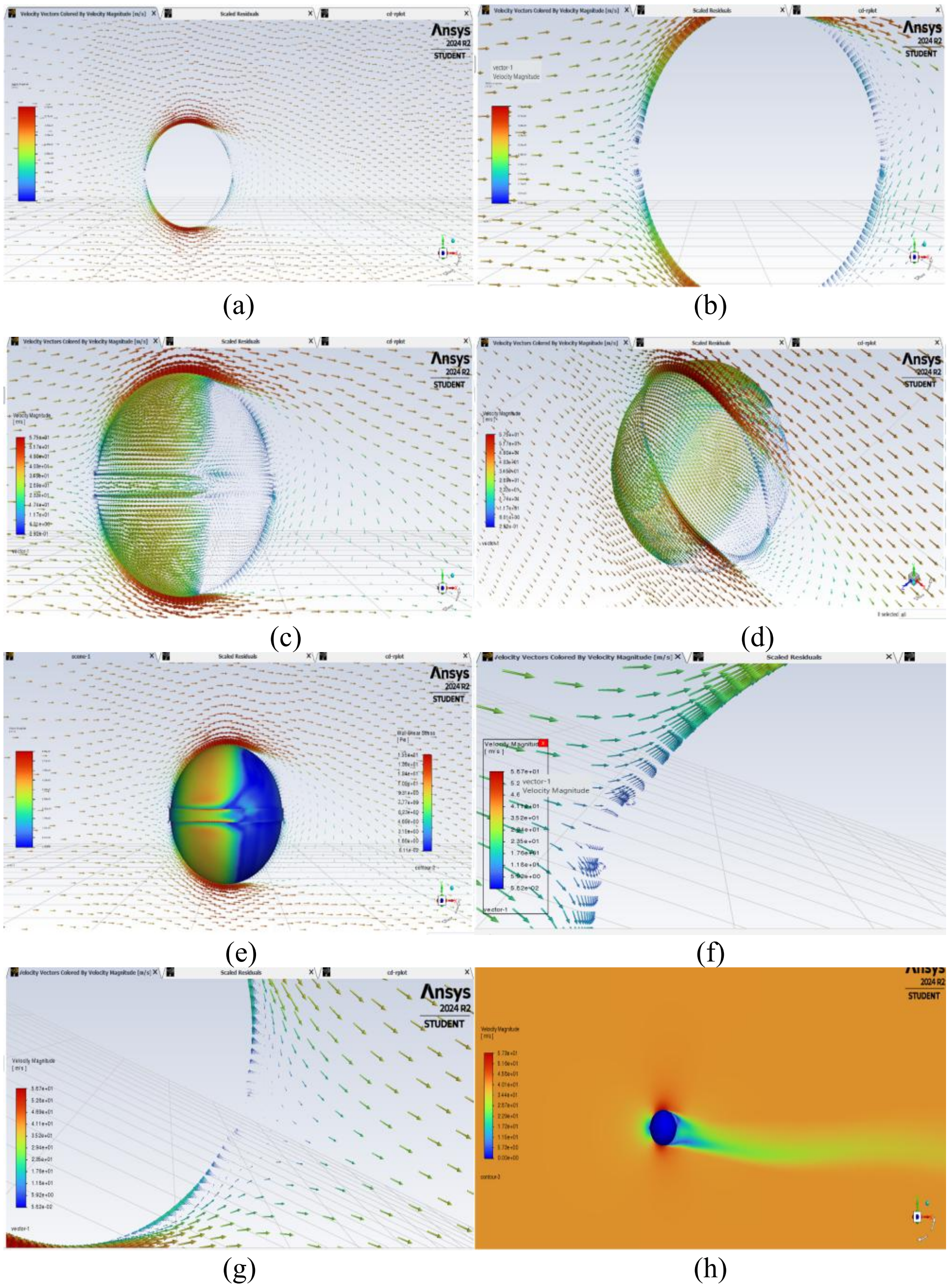


Figure 6. Simulation results in ANSYS Fluent with a velocity of 40 m/s: (a) Velocity vectors and strong deflection, (b) Steep front–rear pressure gradient, (c) Upper flow separation and large vortices, (d) Turbulent asymmetric wake, (e) Low-pressure wake causing swing, (f) High turbulence near separation, (g) Lower flow reattachment, and (h) Elongated drag-dominated wake

This slight deviation from perfect mass balance remains within an acceptable numerical tolerance and does not compromise the validity of the simulation results. The marginally higher imbalance compared to the lower-velocity cases can be attributed to the strongly turbulent, highly irregular vortex structures that dominate at this speed, which inherently increase numerical instability in CFD calculations. Nonetheless, the close agreement between inlet and outlet mass flow rates confirms that the computational setup reliably captures the complex aerodynamic behavior of the cricket ball at high velocity, providing a sound basis for interpreting the corresponding drag, lift, and pressure distribution results.

Flow separation occurs earlier at high speeds, resulting in a greater low-pressure area behind the cricket ball. The flat surface on the ball increases the pressure asymmetry, significantly affecting the swing force. High pressure in front of the ball and low pressure behind create a sharp difference, amplifying the aerodynamic forces at work. At high speeds, the airflow around the cricket ball becomes predominantly turbulent. The flat surface amplifies this asymmetry, resulting in large vortices around the back of the ball.

These vortices create stronger disturbances, increasing the complexity of the airflow around the ball and ultimately affecting its direction and stability. Under these conditions, the drag force reaches its maximum, impeding the ball's motion more than at low or medium speeds. However, the more dominant lift force at high speeds gives the ball more opportunity to swing. This lift force results from the significant pressure difference between the top and bottom sides of the ball, which is amplified by the turbulent flow pattern. The flat-surface effect on the ball causes more pronounced asymmetric disturbances, which are a key factor in generating the swing effect at medium to high speeds. This flat surface triggers a faster transition of the flow from laminar to turbulent, increasing the drag force and enlarging the vortex behind the ball. This phenomenon is consistent with fluid mechanics theory, which states that turbulent flow magnifies aerodynamic forces, making the flow around objects such as cricket balls more complex and dynamic. Figure 7 shows the relationship between velocity and aerodynamic parameters.

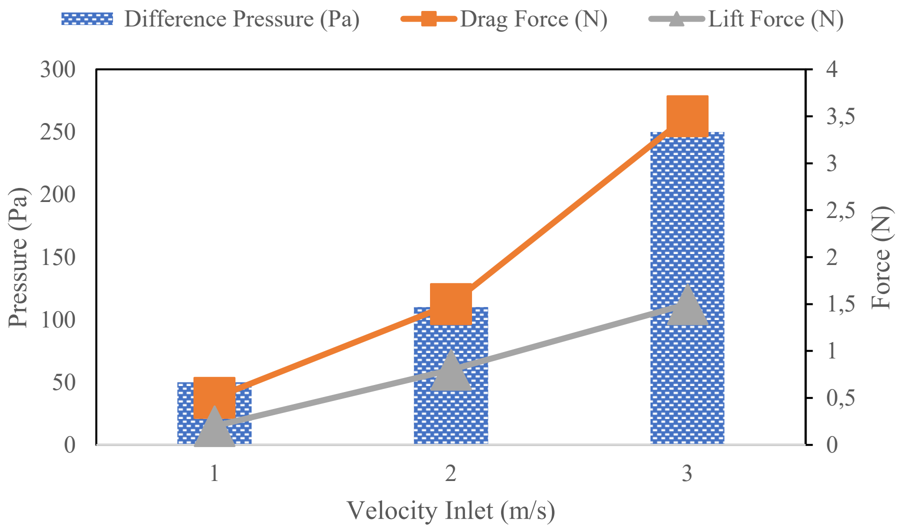


Figure 7. Graph of the relationship between speed and aerodynamic parameters

The drag force experienced by a cricket ball increases exponentially as its speed increases. Drag force is the resistance exerted by the air against a moving ball, and it increases dramatically with increasing ball speed. The drag force is relatively small at low speeds, but as the ball speed increases to medium and high speeds, the drag force becomes much greater. This is because drag is directly proportional to the square of the airflow velocity. Thus, an increase in speed from 20 m/s to 40 m/s results in a much larger increase in drag force than an increase in speed from 10 m/s to 20 m/s. This phenomenon affects the stability and distance the ball can travel in a cricket match. The lift force acting on the cricket ball also increases significantly at higher speeds. The lift force is generated by the pressure difference between the top and bottom sides of the ball, which is caused by the unsymmetrical airflow. The lift force tends to be small at low speeds because the airflow is more stable and laminar. However, this pressure difference at medium and high speeds increases as the airflow becomes more turbulent and vortices become more dominant, resulting in a stronger lift force. This increased lift force enables the ball to swing or turn more sharply, which is highly beneficial for the bowler to deceive the batsman.

The pressure difference between the front and back sides of the ball increases with speed, affecting the overall aerodynamic force. This pressure difference is relatively small at low speeds, as the airflow is more stable and there is little flow separation. However, flow separation occurs earlier at medium and high speeds and is more pronounced, resulting in a larger pressure difference. The high pressure in front of the ball and the low pressure behind the ball create greater drag and lift forces. This pressure difference is crucial to the swing effect, especially at high speeds, where vortices and turbulence prevail. The flat surface of the cricket ball causes significant asymmetric disturbances in the surrounding airflow.

This disturbance triggers a faster transition of the flow from laminar to turbulent, increasing the drag force and amplifying the vortices behind the ball. This is consistent with fluid mechanics theory, which states that aerodynamic forces are magnified by turbulent flow, making the flow around the object more complex and dynamic.

5 Conclusions

This study successfully analyzed the external airflow dynamics around a cricket ball at various speeds using CFD simulations in ANSYS Fluent. The main points of the conclusion are as follows: At low speeds (20 m/s), the flow is typically laminar, and the pressure distribution is relatively symmetrical. High pressure is concentrated in the area in front of the sphere, while low pressure behind the sphere is due to flow separation. At increasing velocities (30 m/s and 40 m/s), the flow transitions from laminar to turbulent. At 30 m/s, the flow pattern shows a significant disturbance around the sphere, which triggers the formation of vortices.

At high speeds (40 m/s), the airflow around the sphere becomes highly dynamic, forming large vortices that amplify the aerodynamic effects. The drag force increases exponentially with speed. At low speeds, the drag force is relatively small and has little impact on the ball's movement. However, the drag force increases dramatically at medium to high speeds due to more turbulent airflow and earlier flow separation. At high speeds (40 m/s), the lift force becomes more significant due to the flow asymmetry amplified by the flat surface of the sphere. This lift force significantly contributes to the resulting swing effect, enabling the ball to turn sharply after being thrown. The pressure difference between the front and back sides of the ball increases with speed. The pressure distribution is symmetrical at low speeds with a slight pressure difference. However, as speed increases, the flat surface of the cricket ball creates flow disturbances that generate vortices and a larger pressure difference. At high speeds, high pressure at the front of the ball and low pressure at the back create strong aerodynamic forces that significantly affect the ball's direction and speed. Simplifying the sphere model by replacing the seams with flat surfaces yields results relevant to the study of aerodynamic effects. The sphere's flat surface can trigger asymmetric flow patterns and amplify the swing effect, especially at medium to high speeds. These flat surfaces make the airflow more turbulent, leading to larger vortices around the sphere, which increase the pressure difference and the lift force acting on it. This suggests that modifications to the ball geometry can be effectively used to simulate different playing conditions and predict ball performance in various scenarios.

The results of this simulation provide essential insights for bowlers on how to utilize pitch speed and ball orientation to create an optimal swing effect. Understanding the airflow dynamics around a cricket ball can help bowlers organize their bowling strategies to outwit batters and increase their chances of taking wickets. For example, at medium to high speeds, bowlers can use the dominant lift force to make the ball turn sharply, making it difficult for batsmen to predict its direction. This research opens up wider opportunities to understand the aerodynamics of cricket balls and improve on-field game strategies. We can explore various playing conditions using CFD simulations and develop new techniques to improve player performance. The results of this study can also serve as a basis for more in-depth follow-up studies on other factors that affect the aerodynamic forces on cricket balls, such as weather conditions, ball wear, and different throwing techniques.

5 Acknowledgement

This paper is the result of research entitled “Implementasi Mesin Hibrida Pengaduk Dan Pencetak Untuk Meningkatkan Kapasitas Produksi Jenang Di Desa Duren, Kecamatan Tugu, Kabupaten Trenggalek, Jawa Timur,“ funded by the State University of Malang through the Program Kemitraan Masyarakat (PKM) scheme with contract number 24.2.43/UN32.14.1/LT/2025.

References

1. C. Lindsay, *Beyond the seam : unveiling the secrets of swing bowling in elite Australian cricket*, Canberra: University of Canberra, 2024.
2. M. Aoyagi, M. Oshima, M. Oishi, S. Kita, K. Fujita, H. Imai, S. Oishi, H. Ohmori, and T. Ono, "Computational fluid dynamic analysis of the nasal respiratory function before and after postero-superior repositioning of the maxilla," *PLoS One*, vol. 17, no. 4, pp. 1-20, 2022.
3. G. Pant and M. S. Dhapola, "Effect of Sports Drinks on Cardiovascular Endurance of Football Players," *Am. Res. Thoughts*, vol. 1, no. 8, pp. 1732-1740, 2015.
4. Y. Trisnoaji, S. D. Prasetyo, M. Choifin, C. Harsito, and A. A. Mahadi, "Enhancing Efficiency in Small-Scale Hydropower: A Comprehensive Review of Archimedes Screw Turbine Design Innovations," vol. 9, no. 1, pp. 1-15, 2025.
5. L. P. Parker, A. S. Marcial, T. B. Brismar, L. M. Broman, and L. P. Wittberg, "Computational Fluid Dynamics of the Right Atrium: A Comparison of Modeling Approaches in a Range of Flow Conditions," *J. Eng. Sci. Med. Diagn. Ther.*, vol. 5, no. 3, pp. 1-11, 2022.
6. M. S. Mauludin, M. Khairudin, R. Asnawi, Y. Trisnoaji, S. D. Prasetyo, S. R. Azizah, and R. T. Wiraguna, "In-depth evaluation and enhancement of a PV-wind combined system: A case study at the Engineering Faculty of Wahid Hasyim University," *Int. J. Power Electron. Drive Syst.*, vol. 16, no. 2, pp. 1274-1283, 2025.
7. V. Resseguier, L. Li, G. Jouan, P. Dérian, E. Mémin, and B. Chapron, "New Trends in Ensemble Forecast Strategy: Uncertainty Quantification for Coarse-Grid Computational Fluid Dynamics," *Arch. Comput. Methods Eng.*, vol. 28, no. 1, pp. 215-261, 2021.
8. S. D. Grimshaw, A. Briggs, and N. R. Atkins, "A review and reassessment of the aerodynamics of cricket ball swing," Flow, vol. 4, article no. E6, 2024.
9. G. X. de Oliveira, S. Kuhn, H. G. Riella, C. Soares, and N. Padoin, "Combining computational fluid dynamics, photon fate simulation and machine learning to optimize continuous-flow photocatalytic systems," *React. Chem. Eng.*, vol. 8, no. 9, pp. 2119-2133, 2023.
10. M. S. Mauludin, M. Khairudin, R. Asnawi, Y. Trisnoaji, S. D. Prasetyo, S. R. Azizah, and R. T. Wiraguna, "Assessing the Technological and Financial Feasibility of PV-Wind Hybrid Systems for EV Charging Stations on Indonesian Toll Roads," *Int. J. Sustain. Dev. Plan.*, vol. 20, no. 1, pp. 291-304, 2025.
11. S. V. Ponnaluri, P. Hariharan, L. H. Herbertson, K. B. Manning, R. A. Malinauskas, and B. A. Craven, "Results of the Interlaboratory Computational Fluid Dynamics Study of the FDA Benchmark Blood Pump," *Ann. Biomed. Eng.*, vol. 51, no. 1, pp. 253-269, 2023.
12. S. S. Kumar, H.V. Prithvi, C. Nandini, "Data science approach to predict the winning fantasy cricket team Dream11 fantasy sports," *Eprint ARXiv, 2022.*
13. Z. Wang, E. R. Galea, A. Grandison, J. Ewer, and F. Jia, "A coupled Computational Fluid Dynamics and Wells-Riley model to predict COVID-19 infection probability for passengers on long-distance trains," *Saf. Sci.,* vol. 147, article no. 105572, 2020.
14. S. D. Prasetyo, Y. Trisnoaji, Z. Arifin, and A. A. Mahadi, "Harnessing unconventional resources for large-scale green hydrogen production: An economic and technological analysis in Indonesia," *Unconv. Resour.*, vol. 6, article no. 100174, 2025.
15. C. Lindsay, R. Crowther, B. Clark, K. Middleton, R. Keegan, and W. Spratford, "Bowler and coach experiential knowledge of new ball swing bowling in elite cricket," *J. Sports Sci.*, vol. 42, no. 2, pp. 146-159, 2024.
16. S. D. Prasetyo, Z. Arifin, A. R. Prabowo, and E. P. Budiana, "The impact of cavity in finned thermal collector on PVT performance using Al2O3 nanofluid," *Int. J. Thermofluids*, vol. 27, article no. 101284, 2025.
17. H. Jordaan, P. S. Heyns, and S. Hoseinzadeh, "Numerical Development of a Coupled One-Dimensional/Three-Dimensional Computational Fluid Dynamics Method for Thermal Analysis with Flow Maldistribution," *J. Therm. Sci. Eng. Appl.*, vol. 13, no. 4, article no. 041017, 2021.
18. L. Assaffat, M. S. Maulidin, Y. Trisnoaji, S. D. Prasetyo, M. A. Rizkita, Z. Arifin, and M. Choifin,

"Improving Grid Stability with Hybrid Renewable Energy and Green Hydrogen Storage: A Study of Karimunjawa Island," *Math. Model. Eng. Probl.*, vol. 12, no. 5, pp. 1524-1534, 2025.
19. R. Crowther, W. Spratford, K. Middleton, and J. Warmenhoven, "A comparison of inswing and outswing bowling arm mechanics in cricket," *Eur. J. Sport Sci.*, vol. 25, no. 6, pp. 1–10, 2025.
20. Y. Trisnoaji, S. D. Prasetyo, M. S. Mauludin, C. Harsito, and A. Anggit, "Computational fluid dynamics evaluation of nitrogen and hydrogen for enhanced air conditioning efficiency," *J. Ind. Intell.,* vol. 2, no. 3, pp. 144-159, 2024.
21. T. Wilberforce, O. Ijaodola, O. Emmanuel, J. Thompson, A. G. Olabi, M. A. Abdelkareem, E. T. Sayed, K. Elsaid, and H. M. Maghrabie, "Optimization of fuel cell performance using computational fluid dynamics," *Membr. (Basel).*, vol. 11, no. 2, pp. 1-21, 2021.
22. D. C. Suman, "Artificial Intelligence in Sport: An Ethical Issue," *Unity J.*, vol. 3, no. 1, pp. 27-39, 2022.
23. S. B. Hussain, S. I. A. Shah, and A. Shahzad, "Optimization and Aerodynamic Design of a Soccer Ball Using Numerical Analysis," in *the 2020 Int. Conf. Eng. Emerg. Technol. ICEET 2020*, Lahore, Pakistan, 2020.
24. A. N. Kapothanillath, S. Pradeep, S. Sathishkumar, and K. Balaji, "The Role of Boundary Layer Theory in Soccer Ball Dynamics," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 1132, article no. 012009, 2021.
25. S. K. Nadar, T. Amrit, S. S. Srinivas, and K. Balaji, "Aerodynamic Characteristics of Baseball and Tennis Ball using CFD Analysis," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 1132, article no. 012023, 2021.
26. S. Iftikhar, S. Sherbaz, H. A. H. Sehole, A. Maqsood, and Z. Mustansar, "Large Eddy Simulation of the Flow Past a Soccer Ball," *Math. Probl. Eng.*, vol. 2022, article no. 455235, 2022.
27. E. Issakhanian, "Numerical investigation of the aerodynamic force variations during rotation of a pitched baseball," *Discov. Mech. Eng.*, vol. 4, article no. 27, 2025.
28. N. R. Atkins, S. D. Grimshaw, and A. Briggs, "The aerodynamics of cricket ball swing: A review of experimental and computational studies," Proceedings of the Institution of Mechanical Engineers, Part P: Journal of Sports Engineering and Technology, vol. 238, no. 2, pp. 123–140, 2024.
29. Y. Morita, S. Rezaeiravesh, N. Tabatabaei, R. Vinuesa, K. Fukagata, and P. Schlatter, "Applying Bayesian optimization with Gaussian process regression to computational fluid dynamics problems," *J. Comput. Phys.*, vol. 449, article no. 110788, 2022.
30. J. A. Badra, F. Khaled, M. Tang, Y. Pei, J. Kodavasal, P. Pal, O. Owoyele, C. Fuetterer, B. Mattia, and F. Aamir, "Engine Combustion System Optimization Using Computational Fluid Dynamics and Machine Learning: A Methodological Approach," *J. Energy Resour. Technol. Trans. ASME*, vol. 143, no. 2, pp. 1-11, 2021.
31. S. Kalburgi, A. Rathi, M. Narayan, L. G. Keni, K. N. Chethan, and M. Zuber, "Computational fluid dynamics study of cricket ball aerodynamics associated with swing," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 75, no. 2, pp. 125–136, 2020, doi: 10.37934/arfmts.75.2.125136.
32. F. Plua, V. Hidalgo, P. A. L. Jiménez, and M. P. Sánchez, "Analysis of applicability of CFD numerical studies applied to problem when pump working as turbine," *Water*, vol. 13, no. 15, article no. 2134, 2021.
33. R. Vinuesa and S. L. Brunton, "Emerging Trends in Machine Learning for Computational Fluid Dynamics," *Comput. Sci. Eng.*, vol. 24, pp. 33-41, 2022.
34. Y. Liu, E. M. Ozbayoglu, E. R. Upchurch, and S. Baldino, "Computational fluid dynamics simulations of Taylor bubbles rising in vertical and inclined concentric annuli," *Int. J. Multiph. Flow,* vol. 159, article no. 104333, 2023.
35. M. Elrefaie, F. Morar, A. Dai, and F. Ahmed, "DrivAerNet++: A Large-Scale Multimodal Car Dataset with Computational Fluid Dynamics Simulations and Deep Learning Benchmarks," in *the Advances in Neural Information Processing Systems 37 (NeurIPS 2024)*, Vancouver, Canada, 2024.
36. Y. Tominaga, L. L. Wang, Z. J. Zhai, and T. Stathopoulos, "Accuracy of CFD simulations in urban

aerodynamics and microclimate: Progress and challenges," *Build. Environ.*, vol. 243, article no. 110723, 2023.
37. P. S. Vineeth and M. V. Panchagnula, "Numerical simulations of a cricket ball trajectory," *Sadhana - Acad. Proc. Eng. Sci.*, vol. 50, article no. 75, 2025.
38. R. Vinuesa and S. L. Brunton, "The potential of machine learning to enhance computational fluid dynamics," *arXiv preprint arXiv:2106.12362, 2021.*
39. M. Mani and A. J. Dorgan, "A Perspective on the State of Aerospace Computational Fluid Dynamics Technology," *Annu. Rev. Fluid Mech.*, vol. 55, pp. 431-457, 2023.
40. F. Pichi, F. Ballarin, G. Rozza, and J. A. N. S. Hesthaven, "An artificial neural network approach to bifurcating flows," *Computers & Fluids,* vol. 254, article no. 105813, 2023.
41. R. Natarajan and Y. Sambath, "Experimental and Computational studies on a cricket ball subjected to various seam angles," 2023. DOI: 10.21203/rs.3.rs-2457547/v1.
42. X. Li and S. C. P. Cheung, "A learning-centred computational fluid dynamics course for undergraduate engineering students," *Int. J. Mech. Eng. Educ.*, vol. 53, no. 2, pp. 256-276, 2024.
43. Q. Cazeres, P. Pepiot, E. Riber, and B. Cuenot, "A fully automatic procedure for the analytical reduction of chemical kinetics mechanisms for Computational Fluid Dynamics applications," *Fuel,* vol. 303, article no. 121247, 2021.