The Role of Boundary Layer Theory in Soccer Ball Dynamics

Abhijith Namboodiri Kapothanillath, Sidharth Pradeep, Shriram Sathishkumar and K Balaji

Department of Mechanical Engineering, Amrita School of Engineering, Amrita Vishwa Vidyapeetham, Coimbatore, India

*k_balaji@cb.amrita.edu

Abstract. The boundary layer theory plays a major role in the development of fluid mechanics applications especially in the sports ball dynamics. Whenever an object travels through the air, it experiences aerodynamic forces acting on it. Sport balls, uses these forces to deviate from its expected path of flight. Thus, creating an advantage which can be exploited by the playing members to improve their skills. The present study focuses on numerical simulation of aerodynamic flow over a soccer ball and estimated the drag coefficient. In order to get more precise results, the effect of gravity is included to study the dynamics of flow. The variation of drag coefficient is plotted as a function of Reynolds number and wind speed. The computational result had a good agreement with existing empirical data.

1 Introduction

Fluid dynamics is the science of movement of liquids and gases. Studying it allows us to understand them and the variety of concepts that come with it. Aerodynamics is a subset of fluid dynamics, dealing with the flow of air. Soccer is one of the most played and loved games in this modern era. In this paper, we discuss the interaction of the ball in flight with the flowing air. The drag coefficient ($C_D$) for different velocities is calculated and validated with experimental values. The aerodynamic properties of a football are a topic of research, which is lagging in its development, as the research carried out in this topic is few.

Experimental analysis done on soccer ball aerodynamics, [5], using a wind tunnel experiment, compared the coefficient of drag for different Reynolds numbers on both rotating and non-rotating balls. A hand-stitched soccer ball and two thermally bonded soccer balls were used for the experiments. A $C_D$ value of 0.43 in the sub-critical region and ~ 0.15 in the super-critical region for a non-rotating soccer ball was obtained. And the critical Reynolds value was in the range of ~ 2.2–3.0 × 10^5. A drag crisis occurring in between the speeds of 12 - 15 m/s was also identified.

The effects of surface roughness on a stationary circular cylinder were discussed and the flow around it was numerically simulated [14]. By running different cases, the effect of surface roughness on $C_D$ value was identified. The drag was very high, around 2.5 times higher than that of a smooth sphere. The effect of lifts was also understood.

The aerodynamic properties of 3 footballs, especially Jabulani, Teamgeist and Fevernova balls of different panel numbers were identified [1]. The $C_D$ values were determined to be 0.15, 0.19 and 0.21 respectively. The Jabulani ball showed a late transition compared to the other two balls. The 32 panel Fevernova ball had an early transition.

The aerodynamics performances of 32 panel (with seams and stitch) and 14 panel (seamless) balls were evaluated [2]. The balls were tested at speeds of 20 km/h to 130 km/h. Their $C_D$ values was...
determined and compared. The 32 panel Nike was found to be better than the Adidas, while it had more panels and an early transition to a turbulent boundary layer, which delayed its separation (100 degrees from stagnation). For the Adidas, due to a lower number of panels, the flow was mostly laminar. The ball acted as a smooth sphere, hence the boundary layer separated early at 90 degrees.

The wake vortex interactions of a curved plate were experimentally evaluated [8]. By changing the chord length to diameter ratio and the angle of flow the generation of wakes was observed for each case. It was inferred from this paper that on a soccer ball the wake region can be deferred due to different velocities. Sometimes, the direction of wind in practical cases (flow angle) would impact the formation of the wake region drastically. The varying of the diameter and sphericity of the ball would also impact the generation of wake.

Recent study on fluid dynamics of soccer ball reported that the number of panels on the ball, their orientation and the changes in the panel shape have a major impact on the aerodynamic characteristic of the ball [9]. Changing the texture of the ball surface would change the aerodynamics of the ball. This was validated with the help of a wind tunnel experiment.

Numerically investigation the aerodynamic characteristic of a square section cylinder at low Reynold numbers was done [7]. The paper has shown $C_D$ values for different corner ratios and concluded that the most effective configuration is of a 0.2 corner ratio which gives a percentage drag reduction of 21.49% along with a very high Strouhal number (0.17).

The rotational and transverse oscillation of a square plate and conducted numerical simulations was discussed [12]. The results were validated using mean drag coefficient for both cases from literature. The generated pressure contours were explained and the correlation between frontal area, vortex patterns and drag coefficient were identified. The influence of Re on $C_D$ was also observed. Thus, this paper looked into the major differences when an object starts to rotate.

An experiment to find out the differences between 4 modern soccer balls and how these differences affected the trajectory of the ball was done [13]. Wind tunnel tests were used to carry out experiments and analyse the four balls. The Adidas Terrapass had optimal sphericity, thereby reducing its form drag. The Nike t90 omni epl had a good design and a panel geometry of hexagons, thereby reducing its drag at low speeds due to an early transition. The Puma v108 had 24 panels and an improved sphericity. The Adidas Terrapass replica is the replica of a Terrapass ball. It was observed that among the four balls, the Puma was better at lower velocities, showing lower $C_D$ values due to an early transition (low critical Reynolds number) and the Terrapass had a late transition, which makes it good at higher velocities (high critical Reynolds number).

A wake is formed downwind of any object exposed to flowing air. The study of this wake can help us to understand various aerodynamic effects such as the drag. In the computational analysis of flow past a curved surface, it was found that by adjusting and introducing curve radius and the orientation of the surface, a $C_D$ reduction of 58% was achieved [3]. The drag produced on a cylinder due to the presence of a notch was computationally analysed and its flow visualisation was studied [4]. The flow was simulated for various angles of the notch with respect to the flow of air. The $C_D$ values were observed to increase till 30° and then decrease till 45° and then finally increase again till 140°.

Most of the research works were experimental and these papers were referred to understand the behaviour of the ball during its flight. But, setting up an experimental set up is not an easy task. Formulating a computational setup is easier. It is observed here, how the coefficient of drag changes with the structure of the ball, surface roughness and the varying velocity and was also able to observe the transition of fluid flow from laminar to turbulent (during critical Reynolds number) for different types of soccer balls.

The two basic equation that are solved are the continuity and the Navier-Stokes equations. The continuity equation is derived from the statement that the mass entering the system is equal to the mass leaving the system. Thus, the continuity equation would be equation 1. As the velocity of air flows never exceeds Mach 0.3, the flow is considered to be incompressible. We also assume that density doesn’t change with time. Hence the continuity equation would eventually become equation 2.
\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0
\]  \hspace{1cm} (1)

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0
\]  \hspace{1cm} (2)

The next equation, Navier–Stokes equation, is actually a set of partial differential equations. This combines the conservation of mass and conservation of momentum equations with Newton’s second law of motion. It is a very sophisticated equation combining the density, pressure, temperature, viscosity, kinetic and potential energies of a fluid. The NS equation is given in equation 3.

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla \cdot (\rho \mathbf{u} \times \mathbf{u}) = -\nabla p + \mu \nabla^2 \mathbf{u} + \rho g
\]  \hspace{1cm} (3)

The Coefficient of Drag (\(C_D\)) is a dimensionless quantity which quantifies the resistance to fluid flow (drag) over a surface. This value is dependent on two types of drag – form or pressure drag and the skin friction drag. The drag coefficient and the drag force can be related by the equation 4.

\[
C_D = \frac{2F_d}{\rho u^2 A}
\]  \hspace{1cm} (4)

The boundary layer is a region of fluid very close to the surface where the effects of viscosity are significant. The flow properties are determined by the boundary layer thickness (\(\delta\)) which is the distance from the solid object’s surface to the fluid whose velocity is 99% of the free-stream velocity. The thickness depends on the viscosity, velocity and the flow profile (laminar or turbulent) – essentially the Reynold’s number \(\text{Re} = \frac{\rho u D}{\mu} = \frac{u D}{v}\). A fluid flow with a very small Reynold’s number will have a large boundary layer thickness compared to the flow with a large Reynold’s number, which will have a small boundary layer. Also, a turbulent flow has a larger boundary layer thickness than a laminar flow. The boundary layer theory bridges the gap between the Euler equation (slip) and the Navier-Stokes equation (no slip condition). Using the Order of Magnitude analysis for the Navier Stokes equations:

The boundary layer equation and the equation for boundary layer thickness can be determined.

**Boundary Layer Equation:**

\[
U \frac{\partial u}{\partial x} + V \frac{\partial u}{\partial y} = U_{\infty} \frac{\partial U_{\infty}}{\partial x} + V \frac{\partial^2 u}{\partial y^2}
\]  \hspace{1cm} (5)

In this paper, our computational drag coefficient, \(C_D\) and drag force was compared with the experimental results obtained [2]. The experimental setup consisted of a ball being attached to the sting with the help of adhesives. The sting had a JR-3 multi axis load cell (a 6 degree of freedom - force and torque sensor). RMIT Industrial Wind Tunnel was used. A computer was used to simultaneously interpolate the 3 forces (lift, drag and side) and their respective moments.

## 2 Methodology

The computational approach is used to solve for the physical equations necessary to calculate the aerodynamics of the flow across the football.

### 2.1 Geometry

**Specification:**

- Ball Diameter : 200mm
- Number of Panels : 32
- Stitches around its surface

### 2.2 Pre-processing

The computations for the flow over a football was done through Ansys Fluent 2020 R2. The geometry for the football was taken from GrabCAD (figure 1). The control volume was extruded over the ball in Autodesk Inventor. The control volume was a cube of dimension 1m. Then Combine was used to
subtract the football from the control volume. This geometry file was imported into Ansys Mechanical to generate the mesh. Due to computational limits, we limited the size of the mesh to 200 mm. This gave the total number of cells to 464821 elements. Figure 2 shows an isometric view of the mesh domain. The figure 3 gives a more closer sectional view of the mesh near the football.

2.3 Setup
After establishing and checking for the mesh quality, now we have to start defining the conditions and set up other solver models which will define the method of solving the physical governing equations. Here, we have used a pressure-based solver as the flow is incompressible and transient state solver was employed to see how the flow field evolves with respect to time. A transient state (un-steady) simulation was done since an unsteady simulation best demonstrates the transition of laminar to turbulent flows and the characteristics of turbulent flows. Hence, transient state simulations were done instead of a pseudo-transient case in a steady state solver. We have chosen a standard k-omega model with standard wall treatment function to account for turbulence in the flow and as already discussed earlier, at high Reynolds number and due to seam orientation, the boundary layer separation reaches past the critical regions and results in turbulent flow after the flow separation point. This model also provides better results for observing free-shear layers and wake regions. Using this model will help in predicting the motion of an object by studying its wake region.

2.4 Boundary Conditions
Boundary conditions are what defines our problem. These conditions are given by us on the exterior of the mesh surfaces as well as to the interior region. This helps the solver to understand what problem we are trying to solve. A slight mistake in the boundary conditions could lead to very different solutions, solutions that may be in its entirety - wrong. The frequently used boundary conditions in Ansys Fluent are inlet, outlet, wall, symmetry and axis among others. Our inlet condition was given to the face facing positive Z-axis and the outlet facing the negative Z-axis. The other four surfaces were given as walls with specified shear as 0. The inlet velocities simulated were 7, 14, 25 and 36 m/s. The turbulence percentage was 5% with a turbulent viscosity ratio of 2. The outlet was given a pressure of 0 Pa. The internal pressure was maintained as 101325 Pa. The boundary conditions used are tabulated in table 1. The figure 4 gives a labelled view of the control volume. These velocities were chosen from the velocities over which the experimental studies were done. The transition point in the experiment was observed between 14 and 20 m/s. The experiment began at 5.55 m/s till 36.11 m/s. Hence the velocities 7, 14, 25 and 36 m/s were chosen.
Table 1 Boundary Conditions

| Solver Type        | Transient Pressure based |
|--------------------|--------------------------|
| Viscous model      | Standard K-omega         |
| Inlet Air Velocity | 20-130 km/h (Along Z axis) |
| Outlet Wall        | 1atm                     |
| Ball Surface       | Specifies shear = 0      |
| Roughness Coefficient | 0.5                  |
| Roughness Height   | 0 m                      |
| Gravity            | -9.81 m/s² (y - axis)   |
| Turbulence intensity | 5%                    |
| Turbulent viscosity ratio | 2                  |
| Control volume dimensions | 1000 x 1000 x 1000 (mm) |

2.5 Solution
After the initial steps, the mesh cells are initialised using hybrid initialisation. In the reference values, the projected area of the ball, its diameter and the velocity were given as reference values in order to aid in the calculation of drag values. The equation was solved by Coupled model, with first order upwinding schemes for solving momentum, turbulent kinetic energy, turbulent dissipation and energy equations. The solution was done for a timestep size of 0.05 s for 10 timesteps. A timestep size of 0.05 s was chosen after experimenting with different timestep size values, taking into account the computational limitations (time taken for simulation). The number of iterations for each timestep was capped at 200 iterations. The residual values for the continuity equation, x, y and Z velocities were given as 0.0001. A report was set up to calculate the Coefficient of Drag and the Drag force along the negative Z axis. The values were also plotted against the time step. The latest time step values were taken into consideration for the results. The boundary layer was fully resolved at higher velocities, since the layer became fully turbulent, thereby delaying the separation. Near the boundary layer, the mesh is uniform.

3 Results
The drag coefficient $C_D$ and drag force acting on the football over different speeds were computed. Values were obtained for 7, 14, 25 and 36m/s. These were validated with the experimental data obtained from [2]. The findings were presented in the graphs. (figures 5 and 6)
The obtained data closely resemble the experimental data. From the graphs (figures 5 and 6), the transition from laminar to turbulent was observed at $Re \approx 200000$, which is close to the experimental value $\approx 200000$. While computing, the $C_D$ value was found to be 0.245 at the transition state. The drag coefficient was found to be maximum at $Re \approx 100000$ which is 0.35 and was minimum at $Re \approx 50000$ with 0.224. The Coefficient of drag decreases as the Reynolds number increases because when it becomes more turbulent the pressure drag will get reduced and at the transition period there will be an abrupt reduction in $C_D$. But after that there will be a slight increase due to the viscous drag.

The computed drag force is closely following the experimental values. This can be explained by the drag force formula where the drag force is proportional to the square of velocity. The maximum variation of the drag coefficient (figure 5) is 0.05 ($\approx 12\%$). It is seen around $Re = 100000$. The maximum variation of the drag force (figure 6) was seen at 130 kmph ($\approx 1$ N) ($\approx 15\%$ change).

From Figures 7 (a), (b), (c), (d) it can be seen that the wake area following the ball is reducing in size as the velocity increases. This is due to the increased turbulence, which delays the flow separation over the ball. This reduces the pressure drag. For 14, 25 and 36 m/s speeds even though the pressure drag is decreasing the viscous drag is increasing which further increases the drag force. The wake for the first three conditions is travelling towards the -$y$ axis. Due to the pressure difference between both ends of the ball, the ball experiences a force along +$y$ direction. This can be seen by its wake travelling towards the -$y$ axis. For the fourth condition (36 m/s), the wake is travelling towards +$y$ direction. This indicated that the ball is experiencing a force in the -$y$ direction.

Pressure reaches maximum at the stagnation point and as it reaches the aerodynamic shoulder pressure becomes minimum and velocity becomes maximum. And as the free stream velocity increases, the stagnation pressure also increases. This can be observed from figure 7 (Velocity contour plots) and figure 8 (Pressure contour plots).

The separation points for each velocity can be observed in the figures 9 (a), (b), (c) & (d). The separation points are progressing inwards towards the wake. For the first 3 cases the separation point is at the top half of the ball. For the last case (figure 9-d), it is at the bottom half. This inversion which occurs at the fourth case can be explained. In the first three cases, the pressure distribution causes the ball to move upwards (We can observe the asymmetric wake progressing downwards).
Figure 7 (a), (b), (c) & (d) - Velocity contours for free-stream velocities of 7, 14, 25 & 36 m/s

Figure 8 (a), (b), (c) & (d) - Pressure contours for free-stream velocities of 7, 14, 25 & 36 m/s
Figure 9 (a), (b), (c) & (d) – Separation point for free-stream velocities of 7, 14, 25 & 36 m/s

Figure 10 (a), (b), (c) & (d) – Pressure distribution on the surface of the football for free-stream velocities of 7, 14, 25 & 36 m/s
In the fourth case (fig. 9-d), the pressure distribution inverts, causing the ball to move downwards and the asymmetric wake progressing upwards. This phenomenon is similar to the contrast swing achieved in cricket balls. Contrast swing is a type of swing which occurs when the seam of a ball is parallel to the direction of flight and the swing is achieved purely based on the difference in surface roughness between the two sides [10]. The orientation of the panels in the football, can cause difference in the overall surface roughness, causing the ball to swing.

The figure 10 shows the distribution of pressure over the surface of the soccer ball. The increase in stagnation pressure at the frontal point of contact of the ball with air can be seen with increasing wind speed as expected. The stagnation pressure is seen to increases from 29.74 Pa to 798.7 Pa for 7 m/s to 36 m/s of wind.

4 Conclusion
The computed results are validated using the experimental data. The Aerodynamic behaviour of the soccer ball was attempted to be understood. The drag coefficient is decreasing with increasing Reynolds number. The drag force is increasing as the velocity increases. With increases in fluid velocity the boundary layer separation point is delayed. The element size was maintained at 200 mm in order to make it computationally feasible as it was a large mesh. Better refinement of the mesh and access to high power computing we can expect to obtain better and more accurate results. Soccer is a very competitive sport today in Europe. Computational studies on the soccer ball can be used to study the dynamics of the ball. These studies can help a player to fully utilize these aerodynamic effects to his fullest advantage. Further study into these aerodynamic effects on the ball can help us better understand this process and implement it in the games.

4 Future study
The following modifications can be studied to further improve the aerodynamic characteristics of the football:
- Footballs with more than 32 panels.
- Curved edges between panels.
- At least 20 small dimples in a panel as in golf balls - this helps in early transition to turbulent flow, leading to delayed boundary layer separation.
- Increasing the sphericity of the ball.
- Rough material can be used for stitching to create turbulence.

References
[1] Alam, F., Chowdury, H., Moria H., Fuss, F.K., Khan, I., Aldawi, F., Subic, A. (2011). Aerodynamics of contemporary FIFA soccer balls. (Elsevier, Ed.) Procedia Engineering, 13, 188-93. doi:10.1016/j.proeng.2011.05.071
[2] Alam, F., Chowdhury, H., Moria, H., & Fuss, F. K. (2010). A Comparative Study of Football Aerodynamics. Procedia Engineering. 2, pp. 2443-8. Elsevier Ltd.
[3] Anil, A., Arunkumar, K., Ajith kumar, R., Hariprasad, C. M., & Mani, T. (2019). Flow around curved plates at low subcritical reynolds number: Investigation of wake characteristics. In Lecture Notes in Mechanical Engineering (pp. 811-9). Pleiades Publishing. doi:10.1007/978-981-13-6416-7_76
[4] Arunkumar, K., & Sarath, R. (2021). Investigation of Flow Structures Around a Circular Cylinder with Notch: A Flow Visualization Analysis. In P. Pant, S. K. Mishra, & P. C. Mishra (Ed.), Lecture Notes in Mechanical Engineering (pp. 181-90). Bhubaneswar: Springer Science and Business Media Deutschland GmbH. doi:10.1007/978-981-15-7779-6_16
[5] Asai, T., & Seo, K. (2013). Aerodynamic drag of modern soccer balls. Springerplus, 2(171). doi:10.1186/2193-1801-2-171
[6] Asai, T., Seo, K., Kobayashi, O., & Sakashita, R. (2007). Fundamental aerodynamics of the soccer ball. *Sports engineering, 10*(2), 101-9. doi:10.1007/BF02844207

[7] Chandar, S. V., Sarath, R. S., & Ajith Kumar, R. (2017). Aerodynamic characteristics of a square section cylinder: Effect of corner arc. *2nd International Conference for Convergence in Technology*, (pp. 814-9). Pune: Institute of Electrical and Electronics Engineers Inc. doi:10.1109/I2CT.2017.8226241

[8] D. Murali, K. H. Balu Haridas, K. Arunkumar and R. A. Kumar, (2017) "Flow past a curved plate: Wake vortex characteristics," *2nd International Conference for Convergence in Technology*, Mumbai, 2017, pp. 318-324, doi: 10.1109/I2CT.2017.8226144.

[9] Hong, S., & Asai, T. (2017). Aerodynamic effects of dimples on soccer ball surfaces. *Heliyon, 3*(10). doi:https://doi.org/10.1016/j.heliyon.2017.e00432

[10] Mehta, R. D. (2008). Sport Ball Aerodynamics. In H. Norstrud, *Sport Aerodynamics* (Vol. 506, pp. 229-331). Vienna: CISM International Centre for Mechanical Sciences, Springer. doi:https://doi.org/10.1007/978-3-211-89297-8_12

[11] Nair, A. S., Kumar, S. A., Kumar, K. A., & Kumar, R. A. (2020). Aerodynamic Characteristics of a Square Cylinder: Effect of Dissimilar Leading Edges. In S. Revankar, S. S., & D. Sahu (Ed.), *Lecture Notes in Mechanical Engineering* (pp. 189-97). Bhubaneshwar: Springer Science and Business Media Deutschland GmbH. doi:10.1007/978-981-15-7831-1_18

[12] Narayanan, B. A., Lakshmanan, G., Mohammad, A., & Kishore, V. R. (2020). Laminar Flow over a Square Cylinder Undergoing Combined Rotational and Transverse Oscillations. *Journal of Applied Fluid Mechanics, 14*(1), 259-73. doi:10.47176/jafm.14.01.30706

[13] Oggiano, L., & Sætran, L. (2010). Aerodynamics of modern soccer balls. *Procedia Engineering, 2*, 2473-9. Elsevier Ltd.

[14] S. Iyer, K, A., R, A., and Dr. Srikrishnan A. R., (2017). Combined Effect of Surface Roughness and Wake Splitter Plate on the Aerodynamic Characteristics of a Circular Cylinder, *IOP Conference Series: Materials Science and Engineering*, vol. 234, p. 012007.