Aerodynamic Characteristics of Baseball and Tennis Ball using CFD Analysis

S Keerthekesh Nadar, T Amrit, S Sanjay Srinivas and K Balaji
Department of Mechanical Engineering, Amrita School of Engineering, Coimbatore, Amrita Vishwa Vidyapeetham, India.

Abstract- The objective of the project is to determine the aerodynamic characteristics of baseballs and tennis balls using Computational Fluid Dynamics (CFD) methods. Most realistic initial and boundary conditions are used to simulate each type of ball and to identify key design aspects that can be applied and modified to enhance the performance of the ball and hence improve the game. The basic conservation equations in fluid dynamics are applied to the domain of study to plot the results. For the inlet, different values of velocity based on the motion of the ball in the game were given. A moderate value of surface roughness was given to model the effects of change in the surface between the two balls and the delay in boundary layer separation subsequently affecting the distance travelled by the ball was seen. By observing the values of lift and drag coefficients we can validate computational results with experimental results obtained from reference journals, thus proving that computational simulation has a place in predicting the trajectory and behaviour of moving balls in real-time. The integration of spin in the simulation also yielded results that showcased the Magnus effect, this visualization is generally not observable on experimental methods since it involves complex processes to simulate spin, hence the trajectory of balls under different shots and pitches has been observed. We further developed an idea that could also be used to determine the trajectory of a dual-axis spinning ball. This would help in building the performance of athletes in the game.

1. Introduction
Baseball and Tennis are the most popular games played across the world. The ‘ball’ is common to both the games and it influence and control the nature of the game under different events. So, to understand the behaviour of the ball and to make the game even more entertaining, the study of aerodynamic characteristics of these different balls are necessary. The study of different balls is quite complex due to the presence of seams, stitches, and other geometrical parameters that directly influences the fluid domain characteristics during different maneuver. So, the player usually tries to predict this nature and changes his game according to the situation. Lift and drag coefficients (Cd and Cl) are two important factors that help us to quantify this behaviour of the ball under different given conditions. The boundary layer formation and pattern directly influence the aerodynamic forces experienced by a body [2]. The orientation of the ball concerning free stream velocity or the tangential direction of the ball movement is very crucial in deciding the trajectory of a ball.

The authors carried out an experimental analysis of baseballs in a wind tunnel to study the drag force at different free stream velocities at four different seam positions [4]. A support system with a sensor to measure force was developed to measure drag force. Another experimental study on similar
grounds showed variation in the flow visualization technique. A novel infrared flow visualization technique was used [6].

Another research is a broader and covers experiments on football, cricket ball, baseball, golf ball, and a tennis ball [10]. Since it was purely experimental, minute details such as boundary layer separation are not accurately found. There are two types of pitching techniques in baseball; one is by spinning the ball while pitching and another with almost zero spins which is also known as a knuckleball. In another work the experimental results of the coefficient of drag to influence the design of his project to attain optimum performance is studied [7].

The differences in trajectory caused by spin on the tennis ball by pitching it against a smooth ball and observing the results is seen in [7]. The experiments are carried out in a wind tunnel in the range of 80,000 < Reynolds number < 250,000. To create spin, the ball is suspended by a 3.5 mm diameter string and rotated close to 3000rpm. A similar procedure is followed in the numerical analysis conducted by [8], where he studies the boundary layer development and wake structures at different L/D ratios of a cylinder. Whereas [13] study the variations in the coefficient of drag (Cd) over a cycle as well as the time-averaged value of Cd of different L/D ratios. Further study gives an idea about fluid flow over a body at a low Reynolds number and its corresponding aerodynamic performance [12].

The study that [1] did begins by analysing the flight trajectory equations and balancing the gravitational force with the centripetal force of the spinning ball and the aerodynamic forces of drag and lift. Since all the results were those of experimental studies, it allowed verification of the computational results with data from the experiments.

2. Methodology
For this study, we have used the numerical approach for solving the physical governing equations. In the case of ball aerodynamics, we mainly solve for two governing equations which are continuity equation for incompressible flow (as the flow over a ball is always incompressible) and the Navier’s Stokes equation. These two equations help us in predicting the flow field passing over a ball during its flight. But solving those analytically is almost impossible due to the complex design of balls. So, only numerical or experimental methods can be equipped to solve these kinds of problems. Model to study the variation in aerodynamic efficiency by providing incremental values of Reynold’s number and iterating is studied in [11]. On the other hand, [14] use a method of study which extracts results based on the variation of the position of the body in the study. Due to the lack of time, resources, and facility, we have decided to perform a few computational fluid dynamics simulations at different conditions and tried to understand the dynamics behind it to correlate with experimental results and establish the usage of CFD for predictive purposes based on the accuracy of results.

2.1. Geometry and Pre-processing
The model has been approximated by neglecting the stitches due to involvement of complex meshing and processing requirements. Therefore, we used roughness height equal to the stitches thickness. The seam of the tennis ball is neglected during analysis as previous experimental research papers have proven that the aerodynamic effects of seam are negligible on a tennis ball moving at a speed of 25m/s. For meshing, analysis, and extraction of results, Ansys Workbench and Fluent software are used. A fluid domain is created that encloses the ball. This fluid domain will represent the region of free stream airflow over our ball. So, this is our domain of study. Before further proceeding, the ball needs to be removed out of the domain and captured as space which the fluent software further assumes as the walls of the ball. To do that we will be using a Boolean operation that subtracts the ball from the fluid domain. So, all the surface features are traced in the fluid domain. Then it becomes just one integrated structure with all the surface features engraved in the fluid domain. Few of these techniques and methods have been developed from [9].
Table 1. Domain & mesh specifications

|                     | Baseball                | Tennis ball            |
|---------------------|-------------------------|------------------------|
| Ball diameter       | 72mm                    | 65mm                   |
| Computational domain| Cylindrical             | Cuboidal               |
| Domain dimensions   | Radius = 250 mm & Height = 1250 mm | 1000 mm x 800 mm x 800 mm |
| Mesh size           | 3 mm                    | 3 mm                   |
| Mesh shape          | Tetrahedron             | Tetrahedron            |

Figure 1. Grid independence test plot

This plot shown in Figure 1. describes the change in Cd values as the number of elements is gradually increased. Results had been extracted for 9 different mesh sizes in the range of 3 – 100 mm. It was observed that as the number of elements approached 3 mm the slope of the plot remained constant with increase in the number of elements henceforth exhibiting grid independence.

2.2. Setup

After establishing and checking for the mesh quality, initial conditions are defined and solver models are set up to establish the method of solving the physical governing equations. A pressure-based solver is used as the flow is incompressible, and a transient state solver was employed to see how the flow field evolves concerning time. Further a standard k-epsilon model with a standard wall treatment function is chosen so as to account for turbulence in the flow because this model is generally good for external flow over complex geometry, has a good convergence rate, and has relatively fewer memory requirements. Other models generally do not converge at faster rates. In our case low-pressure gradients are observed, so k-epsilon is better for these situations. And the scope of our study lies in regions beyond the walls as well, in these cases balls pitching characteristics under different rotating conditions were studied and it is observed how streamlines are like when a ball moves in a time frame. So, keeping all these conditions in mind we chose to go with the k-epsilon turbulent model for our simulation as also indicated by [9] in his research work. Standard k-epsilon models are efficient for high Reynolds number flows and can be beneficial in the abstraction of data in flows that have rotational components in them. Finally, when compared to realizable and RNG model, a standard k-epsilon model is more stable. At high Reynolds number, 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.
2.3. **Boundary Conditions**

2.3.1. **Inlet:** This is the region where the fluid (here, air) enters the domain. So, we have given the velocity inlet boundary condition in this region. The velocity defined was normal to the boundary which is along the negative direction of the Z-axis. The turbulence at this boundary was specified as having 1% intensity and with a 0.5 m of length scale. One thing we have to keep in mind is that here, the fluid has motion instead of the ball because the effect caused by either of the aspects is the same.

2.3.2. **Ball surface:** This is the region where the ball interacts with the flowing fluid. So, for Navier’s Stokes equation at this region, we will define the no-slip condition at the wall. For simulating knuckleball conditions, we will be having a stationary wall with the no-slip condition but for curve balls and other spinning ball conditions, we will have to define the no-slip condition with moving walls.

2.3.3. **Outlet:** In this region pressure boundary condition is chosen over velocity boundary condition keeping in mind that the velocity characteristics of the fluid stream after passing over the ball cannot be predicted. We know that the outlet pressure will be at 1 atm considering the outlet to be in an ambient environment, and all other turbulence parameters are further defined by inlet conditions.

2.3.4. **Fluid domain wall:** This is the outermost region of fluid flow. Here, we will define a zero-shear boundary condition with a stationary wall. We define zero shears because the effect of this wall shouldn’t be affecting the fluid flow and it has been checked that this wall lies at the inviscid region of flow, which means the viscosity effects are almost negligible at this region.

| Table 2. Simulation specifications of baseball and tennis ball |
|-----------------------------------------------------------|
| **Baseball** | **Tennis Ball** |
| Solver type | Transient Pressure based | Transient Pressure based |
| Viscous model | Standard k-epsilon | Standard k-epsilon |
| Inlet air velocity | 8 – 39 m/s (along X-axis) | 25 m/s (along X-axis) |
| Outlet pressure | 1 atm = 101325 Pa | 1 atm = 101325 Pa |
| Outer wall | Specified shear = 0 Pa | Specified shear = 0 Pa |
| Ball surface | Moving wall: 15 - 70 rev/s (Spin) | Moving wall: 50 rev/s (Spin) |
| Roughness coefficient | 0.3 - 0.8 | 0.1 |
| Roughness height | 0.762 mm | 2 mm |
2.4. Solution
After setting up all the boundary conditions now we have to set up all the solver settings which will help us to solve our problem according to the input conditions. Initialization plays an important role in solving the governing equations, so here we have used hybrid initialization for more realistic initialization. It simplifies the process of computing values, and we have used coupled model and second-order upwind schemes for solving momentum, turbulent kinetic energy, turbulent dissipation, and energy equations. These solver settings were cross verified with research paper [9] before employing them.

2.4.1. Case at which the baseball spins in counter clockwise direction about X-axis or this type of pitch is also known as topspin

![Figure 3. Velocity contour of base ball](image)

![Figure 4. Boundary layer near stitches of base ball](image)

In this case the baseball has been numerically simulated for the topspin condition. The boundary layer separation at the ball surface can be visualized in Figure 4. And the wake region and the velocity distribution can be identified from the Figure 3. For, the topspin condition the Figure 4 also shows how the seam and stitches induces a non-uniform boundary layer and which leads to prior flow separation.

2.4.2. Case at which the baseball spins in clockwise direction about X-axis or this type of pitch is also known as backspin

![Figure 5. 3D streamline representation](image)
The high-pressure region at the bottom of the ball and low-pressure region at the top of the ball lead to lift. And due to this we can clearly see the lift in the baseball or let's say the positive magnus effect in Figure 5.

2.4.3. Numerical simulation over baseball for different pitches.
The simulation has been performed to replicate a few pitching styles; the extracted results give us better visualization for these types of pitches. The following are few types of common pitches using in the baseball game.

In this case the baseball rotates about two axes, here it rotates about X and Y axes. The Figure 6a and 6b shows the deflection in the baseball path which has been caused due to spinning of the ball.

In this case the baseball rotates about the Y axis and slides towards the right direction. The Figure 7a and 7b shows the deflection in the baseball path which has been caused due to spinning of the ball in the Y axis.

In this case the baseball rotates about the Y axis and slides towards the right direction. The Figure 7a and 7b shows the deflection in the baseball path which has been caused due to spinning of the ball in the Y axis.
In this case the baseball topspin or say spins about the Y axis and starts to sink downwards. The Figure 8a and 8b shows the deflection in the baseball path which has been caused due to spinning of the ball in X axis.

![Figure 9a. Front view of a Gyroball](image)

![Figure 9b. Isometric view of a Gyroball](image)

In this case the baseball rotates about the Z axis, we can see in Figure 9a and 9b that there is no change in the flight path of the baseball as it travels straight towards the batter. This pitch is unique than other spins as there is almost no lift in the baseball and no magnus effect seen in this scenario.

2.4.4. Tennis ball. Net drag along X axis:

Tennis Ball = - 0.402 N (Spinning case) & Tennis Ball = - 0.72 N (Non-spinning case)

This result conclusively shows that spinning balls have lesser drag when compared to non-spinning balls. The negative sign in these values indicate the direction of the force being applied due to spin effects.

![Figure 10. Streamlines in the form of tubes](image)

Figure 10 shows streamlines from the surface of the inlet to determine velocity magnitude and direction at different points across the ball in both X-Y & X-Z plane. Deviation from a straight path explains the presence of Magnus force. This is the velocity contour in the X-Z plane. As we can see the velocity is greater at the bottom than at the top which eventually creates a pressure difference.
Figure 11a. Front view  
Figure 11b. Side view  

Figure 11a depicts how the spin occurs concerning two axes and as it can be observed the velocity is maximum when the vector is pointed towards the flow direction (-X-axis) and minimum when pointed against it. In Figure 11b the wake region seen behind the ball has a green shade indicating the velocity of air to be lower than that of free stream velocity and stagnation region, when seen closely, would approach a color close to dark blue indicating zero flow velocity.

3. Results  
3.1. For base ball, few of the simulation results were validated by comparing these results with that of the experimental results which have already been performed by other researchers. The experimental result pictures were taken from the research paper [10].

3.1.1. At flow speed of 21 m/s and the baseball rotates at a counterclockwise direction at a speed of 1.5 rev/sec

Figure 12a. Smoke photography [10]  
Figure 12b. Numerical simulation result

3.1.2. At flow speed of 21 m/sec and the baseball stays stationary with no spin.

Figure 13a. Smoke photography [10]  
Figure 13b. Numerical simulation result
Table 3. Baseball results comparison

|                        | Cd   | Cl   |
|------------------------|------|------|
| Simulation results     | 0.317| 0.113|
| Experimental results   | 0.3 - 0.4 | 0.1 - 0.2 |

The experimental results were compared with simulation results for a particular range of spinning speed and free stream velocity. The simulation results were cross verified with [10] experimental results.

3.2. Tennis ball
To verify the results obtained a few reference papers have been chosen based on their experimental validation procedure. In the paper [3], it was estimated that the maximum lift coefficient acting on the ball is approximately 0.54 moving at speed of 25m/s at the maximum spin rate of 3000 rpm. In the paper [5], Tests for the spinning conditions (250–2750 rpm) were conducted at wind speeds of 25 and 50 m/s, and data for the new tennis ball (with sufficient roughness) was observed to have drag coefficients in the range 0.6–0.7. These conclusively prove that the results obtained through simulations are genuine values and that these values can further be used to plot the 3D trajectory of a tennis ball.

Table 4. Tennis ball results comparison

|                        | Chadwick | Haake SJ |
|------------------------|----------|----------|
|                        | Cd       | Cl       | Cd*      | Cl*      | Cd* | Cl*      |
| Along X axis           | 0.657    | -        | -        | -        | 0.6 - 0.7 | - |
| Along Y axis           | -        | -0.338   | -        | 0.54     | -    | -        |
| Along Z axis           | -        | -0.337   | -        | -        | -    | -        |

4. Future prospects
A spinning tennis ball launched in the 1st serve, with a dual axis spin, experiences the following forces: Frontal drag force (along X axis), Axial lift force (spinning along Y axis), Lateral lift force (spinning along Z axis), Gravity (along Y axis)

Figure 14. Trajectory representation
The Figure 14 shown is a pictorial representation of the trajectory a tennis ball simulated under dual spin condition. The combined equations of motion of ball and fluid dynamics can be iterated using the Runge-Kutta method. This method of plotting the trajectory by balancing forces along each axis using MATLAB simulations can be used for all types of sports balls. Finally, to induce dual axis spin as a part of experimental validation a gyro setup can be used.

Position and Time:

\[ x = \frac{1}{g} \int_0^\tau \frac{v^2 \cos \tau}{\cos \tau + M^*} d\tau \]  
(1)

\[ y = y_0 - \frac{1}{g} \int_0^\tau \frac{v^2 \sin \tau}{\cos \tau + M^*} d\tau \]  
(2)

\[ z = -\frac{1}{2} a_z t^2 \]  
(3)

\[ t = -\frac{1}{g} \int_0^\tau \frac{v}{\cos \tau + M^*} d\tau \]  
(4)

Force along Y:

\[ m \frac{v^2}{R} = mg \cos \tau + M \]  
(5)

Force along X:

\[ m \frac{dv}{dt} = -D - mg \sin \tau \]  
(6)

Force along Z:

\[ m \frac{v^2}{R} = L; \quad a_z = \frac{L}{m} \]  
(7)

\[ \frac{dv}{dt} = \frac{\sin \tau + D^*}{\cos \tau + M^*} v \]  
(8)

\[ D^* = C_D (\pi d^2 / 8mg) \rho v^2 \]  
(9)

\[ M^* = C_L (\pi d^2 / 8mg) \rho v^2 \]  
(10)

Where,

\[ D^* = D/mg; \quad M^* = M/mg; \]

M = Axial lift force (Magnus force)

L = Lateral lift force (Magnus force)

D = Frontal drag force
5. Conclusion

Numerical simulations on both baseball and tennis balls have been carried out for predicting their aerodynamic characteristics such as lift and drag coefficient. Streamlines and contours of pressure and velocity are extracted for the domain nearest to the ball and have been visualized for required conditions. Finally, these results were compared and validated with the known experimental data which have already been published by other researchers. Future prospects of the study have also been discussed at the end of the study.

References

[1] Antonin S, 1988, The aerodynamics of tennis balls—The topspin lob, Am. J. Phys. 56,138: 10.1119/1.1569
[2] Cengel Y, Simbala JM, Fluid Mechanics - Fundamental and Applications 4th Edition, McGraw Hill Education
[3] Chadwick, Stephen George, 2003, The aerodynamic properties of tennis balls. PhD thesis, University of Sheffield.
[4] Firoz A, Huy H, Harun C, Aleksandar S,2011, Aerodynamics of baseball
[5] Haake SJ, Goodwill SR,Carre MJ, 2007, A new measure of roughness for defining the aerodynamic performance of sports balls
[6] James A. Scobie, Carl M. Sangana, Gary D. Locka, 2014, Flow visualisation experiments on sports balls, The 2014 conference of the International Sports Engineering Association Procedia Engineering 72 738 – 743
[7] Kumar, T.R.S., Venugopal, S. Ramakrishnananda, B., Vijay, S., 2020, Aerodynamic performance estimation of camber morphing airfoils for small unmanned aerial vehicle, Journal of Aerospace Technology and Management Volume 12, Issue 1, Article number e1420
[8] Pillai, S. M., Arun Kumar, K., & Ajith Kumar, R. (2019). Flow around a circular cylinder: Effect of twin splitter plates. International Journal of Mechanical and Production Engineering Research and Development, 9(2), 838-47.
[9] Pouya J, P Kreun, Hady M, William WL,2014, Computational Aerodynamics of Baseball, Soccer Ball and Volleyball, American Journal of Sports Science 2(5):115-121
[10] Rabindra M, 1985, Sports Ball Aerodynamics Vol.17:151-189
[11] Rajendran, A.K.a, Shobhavathy, M.T.b, Kumar, R.A.a, 2015, CFD analysis to investigate the effect of vortex generators on a transonic axial flow compressor stage, ASME 2015 Gas Turbine India Conference, India; Code 119665
[12] Rajesh, S.K.T. Sivakumar, V., Ramakrishnananda, B., Arjhung, A.K., Suriyapandiyan, 2017, Numerical investigation of two element camber morphing airfoil in low Reynolds number flows, Journal of Engineering Science and Technology, Volume 12, Issue 7, Pages 1939-1955
[13] Rajesh, R., & Rakesh, S. G. (2020). Effect on the drag coefficient of various spiked cylinders during buzz phenomenon subjected to hypersonic flows. Journal of the Brazilian Society of Mechanical Sciences and Engineering, 42(6). doi:10.1007/s40430-020-02384-5
[14] Sudhamshu, A.R., Pandey, M.C., Sunil, N., Satish, N.S., Mugundhan, V., Velamati, R.K., 2016, Numerical study of effect of pitch angle on performance characteristics of a HAWT, Engineering Science and Technology, an International Journal, Volume 19, Issue 1, Pages 632-641