Case Study

Effect of side spin on a football (soccer ball) using Ansys Fluent

Gautham Varma Raja Kochanattu and Heyang Zhang

Developed and curated by the Ansys Academic Development Team

education@ansys.com

Summary

Ansys Fluent is a Computational Fluid Dynamics (CFD) software that is used to solve various problems related to fluid flow, heat and mass transfer, chemical reactions, and more. It uses advanced physical models like turbulent modeling, multiphase modeling, battery modeling, combustion, and fluid-structure interactions to solve the given problem to high level of accuracy.

In this case study, Ansys Fluent is used to understand the effect of spin on a football (soccer ball in North America) using turbulence modeling. Bernoulli's principle will be used to explain the curvature of the ball. The description of boundary layer separation as well as the turbulent wake formation will be visualized using the Fluent post-processing tool.

Table of Contents

1. Introduction

Football (referred to as Soccer in North America) is one of the most popular sports in the world, with the finals of the World Cup viewed by a whopping 3.575 billion people across the globe. The ball is the most important part of the sport, with the dynamics and the shape and size of the ball play an important part in the skillset. The Football association (first established in 1863) set the rules for the size and shape of the ball in 1872. Based on the rules, the ball should be:

- Spherical
- Circumference between 68.6cm-71.1cm
- Weight between 400g-440g
- Pressure of the ball between 0.6atm-1.1atm at sea-level.

Earlier versions of the football were made of pig's bladder. However, later with the introduction of an inner bladder, the outside was made of leather stitched together. Initially, there were six panels that were stitched together. In this case study, the football being simulated is based on the iconic Adidas Telstar which was introduced during the 1970 Mexico World cup (Figures 1 & 5). This ball hads20 hexagonal and 12 pentagonal surfaces sewn together to form the outer surface. Fluid dynamics simulation can help us understand these changes as well as the change in the roughness of the ball while being kicked through the air. This case study takes one such example where there is a side spin of the ball and its effect on the movement of the ball and Ansys Fluent to simulate the problem.

Figure 1: A standard Adidas Telstar soccer ball being kicked

2. Problem Statement

Freekick is a very important method in football by which the team is given an opportunity to advance the ball into the opposition's half or score a goal. There are different techniques to used by players during a freekick, the main of which is a curve ball. In this study, we will run two simulations, one where the ball is not spinning and the other with a side spin to understand how the fluid dynamics is used by fluent to predict the movement of the ball in the air. We will be using Telstar football to run the analysis. We will also try to understand the turbulent flow around the football and how the specific shape and structure of the football might help in stabilizing the football.

3. Physics Behind the calculations- Bernoulli's Principle

Let us consider an incompressible, isothermal and constant viscosity (*μ=const*) flow for which the Navier's Stokes equation is:

$$\nabla \cdot \vec{V} = 0$$

$$\rho\left[\frac{\partial u}{\partial t} + \nabla \cdot (\vec{V}u)\right] = \frac{\partial P}{\partial x} + \mu\nabla^2 u + F_{b,x}$$

$$\rho\left[\frac{\partial v}{\partial t} + \nabla \cdot (\vec{V}v)\right] = \frac{\partial P}{\partial y} + \mu\nabla^2 v + F_{b,y} \quad (1)$$

$$\rho\left[\frac{\partial w}{\partial t} + \nabla \cdot (\vec{V}w)\right] = \frac{\partial P}{\partial z} + \mu\nabla^2 w + F_{b,z}$$

The above equation reasonably explains most low speed fluid flow problems and is used to solve a wide range of problems of fluid dynamics. For an incompressible fluid, in addition to constant density and viscosity, and assuming that the thermal conductivity (k) and specific heat (C_p) are constants, a simple Energy equation can be obtained.

$$\rho C_p\left[\frac{\partial T}{\partial t} + \nabla \cdot (\vec{V}T)\right] = k\nabla^2 T + (\overline{\overline{\tau \cdot \nabla}})\vec{V} + \dot{S}_g \quad (2)$$

Here ρ is the density, $\vec{V}$ is the velocity vector given by $u\hat{i}+v\hat{j}+w\hat{k}$, P is the pressure, $F_{(b,i)}$ and $\dot{S}_g$ are the body forces, and T is the temperature. The equations are decoupled for non-isothermal flows; thus, the velocity can be solved separately using the incompressible N-S equations and can be used to update the temperature of the flow.

Bernoulli's Principle is a more restrictive version of the conservation of energy principle of the fluid flow. The following restrictions/ simplifications are considered:

- Steady-state flow
- Incompressible irrotational flow
- No heat is transferred to or from the fluid.
- No work is done on or by the fluid.

These assumptions can be applied to the above equations and the following is obtained:

$$\nabla\left(P + \rho\frac{V^2}{2} + B\right) = 0 \quad (3)$$

In the above equation, for the gradient to be zero, the term inside the parentheses must be a constant and hence:

$$P + \rho\frac{V^2}{2} + B = Const. \quad (4)$$

This is the most common form of Bernoulli's equation where the three terms of the equation are:

- P= Internal Energy
- $\rho(V^2/2)$= Kinetic Energy
- B= Potential Energy

Thus, Bernoulli's equation can be physically represented as the **Conservation of Energy** for a steady-state, incompressible, irrotational fluid flow.

Since the total energy for the problem is constant everywhere, it can be calculated by values at far field which can then be compared with other points on the flow fields to obtain the various variables downstream (Figure 2).

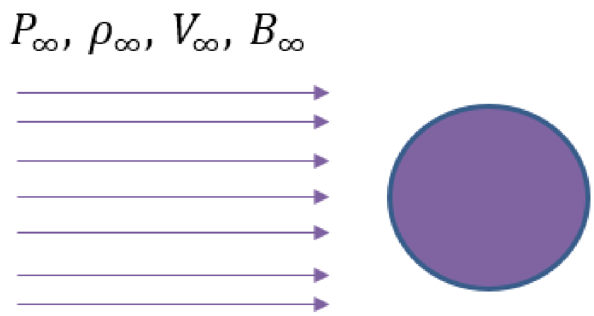

Figure 2: Figure depicting the flow characteristics at Far Field

$$P_\infty + \rho_\infty \frac{V_\infty^2}{2} + B_\infty = Const. \quad (5)$$

If we neglect the body forces effect from the above equation (Because, in our example, we do ignore the effect of body forces like Gravity), we end up with the equation of total Pressure:

$$P_{tot} = P + \rho \frac{V^2}{2} \quad (6)$$

Where,

P is the **static pressure** and

ρ ($V^2/2$) is kinetic energy term but has pressure dimensions and is called the **dynamic pressure**.

Equation 6 suggests that if there is a velocity difference between two points in a fluid flow, it leads to having a force acting in the direction of the higher velocity region. A detailed explanation and derivation of the Bernoulli's Principle can be studied in the Ansys Innovation Course titled Simple Approximation of Fluid Flows.

The most classic and famous example of this phenomena is utilized to calculate the Lift force of an airfoil. Consider an airfoil as described in Figure 3. When we consider two points in upstream and downstream of the airfoil, the total mass flowrate should be same (conservation of mass). Because of this, the air moving above the airfoil will have to travel a longer distance than the air moving below for the same time. This suggests that the local velocity of the flow above is higher than the local velocity below the airfoil ($v_2>v_1$). If we consider Equation 6, the pressure on the top of the airfoil is lower than the pressure on the bottom of the airfoil. This leads to a net force acting on the airfoil in the upward direction generating the lift.

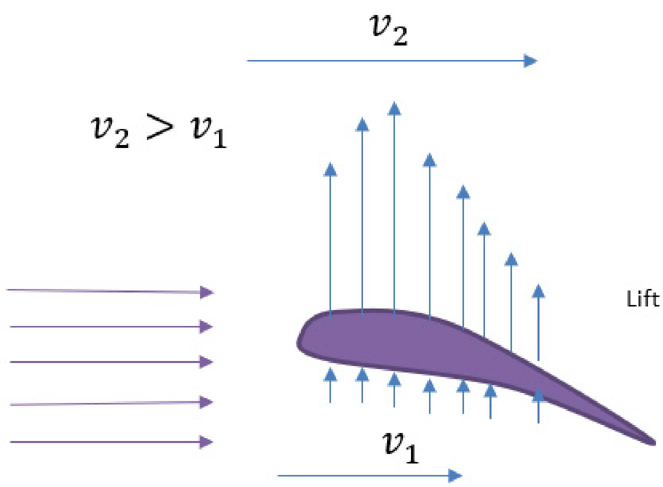

Figure 3: Flow around an Airfoil

This Lift force can be calculated by the formula:

$$F_L = \frac{1}{2}\rho(v_2^2 - v_1^2) \tag{7}$$

Like the airfoil example, a spinning ball is considered moving in air mimicking a freekick being taken during the game of football (Figure 4). While the ball is spinning, a small boundary layer (of air) close to the surface of the ball is also spinning in the same direction as the ball. The figure shows that this boundary layer is flowing in the same direction of the flow on one side of the ball while it is moving in the opposite direction on the other side of the ball. Because of the impact of this spinning boundary layer, the average flow velocity where the boundary layer meets the freestream will be slightly higher or lower than the freestream velocity respectively. According to Bernoulli's principle, the sides will now have a pressure imbalance which leads to a force being acted on the object, pushing the object in the direction of higher velocity. This phenomenon by which there is a bending of the spinning ball in motion through air is called the **Magnus effect** (Figure 4). These can be seen in various sports like Football (bending of the ball during freekick or shot), cricket (drifting of the ball during spin) *etc.*

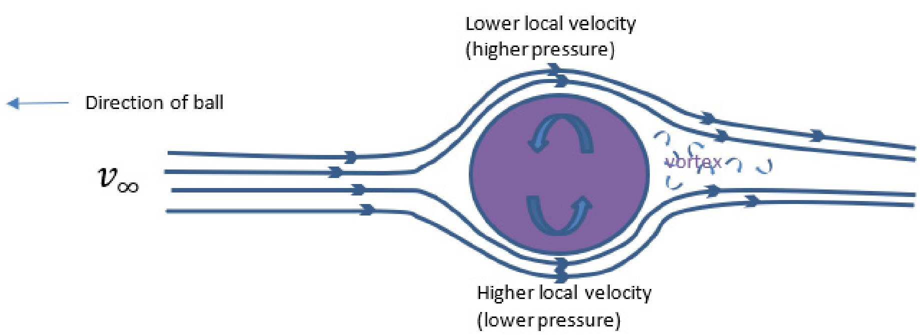

Figure 4: Flow around a spinning ball

4. Geometry and Meshing

The CAD file for the Telstar football was designed with a simple truncated icosahedron structure (Figure 5 left). The size of the ball is 114.44mm which is equivalent to the rules that was stated in the previous section. The football structure is imported and an enclosure of size 10×5×5 m^3 was used to define the wind tunnel using Ansys Discovery 23R1. Such a big enclosure is utilized so that there are no interferences from the wall of the enclosure to the actual movement of the ball. A volume mesh was generated using the Watertight Geometry Workflow (for more detailed meshing technique please follow the link) using the Ansys Fluent Meshing. During the meshing, four different named sections have been introduced (inlet, outlet, Wall, and ball). A good volumetric mesh was achieved with maximum skewness of 0.404 by having very fine mesh near the ball in order to capture the structure of the ball (Figure 5 right). A detailed mesh sensitivity study was carried out. This study is a technique that involves running the same simulation using grids with different resolutions and check the accuracy of the results. This is a very important step in simulation to constructing a mesh that will give us an accurate solution with maximum computational efficiency. With the mesh sensitivity technique, the mesh with a total cell count of 1,367,252. This will give a reasonably accurate value for the forces acting on the ball while spinning efficiently.

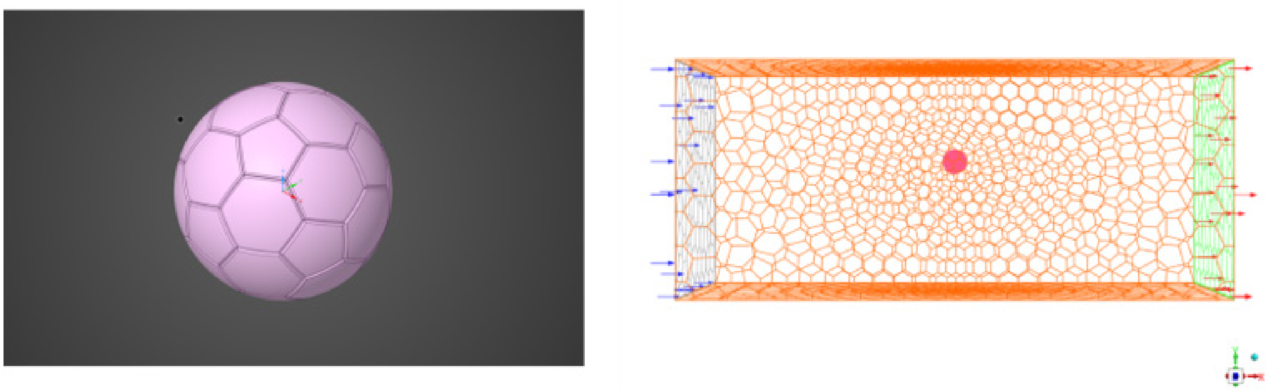

Figure 5: The Telstar Football with the truncated icosahedron structure (left). The Mesh generated using the Watertight Geometry Workflow (right)

The spin on the ball is defined under the boundary conditions in the Ansys Fluent workflow. The Ball is considered as a wall with motion activated to rotation about the Y axis. The speed of rotation is 50 rads/s and the inlet velocity of the fluid (air) is 25 m/s. A transient Pressure based calculation is carried out in Ansys Fluent using Realizable K-ϵ turbulence model.

5. The Results

The calculation is run for 20 timesteps with each time step having a size of 0.001s. In our calculation, we assume that the football is at its maximum height and translational speed and stays in this value only for an average of 0.12s. The calculation is carried out at the highest point of trajectory of the ball; hence the body forces can be ignored.

Table 1: Force acting on the football vs the spin speed of the ball.

Spin on the Ball (rads/s)	Force acting in the Z direction (N)
0	0
25	2.19617
50	3.310064
75	3.947636
100	4.370827
150	4.98569
200	5.412142

Since the ball is spinning about the Y axis while moving in the X direction, a total force should be exerted in the Z direction according to Bernoulli's Principle. For a speed of 50 rads/s, it was seen that the Force acting in the Z direction was 3.310064 N. The same calculation was carried out for different speeds of spin for the football (defined in Table 1) and was found that as the spin speed increased, the forces acting in the Z direction also increased. This is congruent with the principle as the more the difference in the local velocities occur, the higher the values of the Force acting on the ball. When these values are plotted (Figure 6), the Force acting upon the ball is seen as increasing with increase to the speed of spin on the ball as described by Briggs (Briggs, 1959).

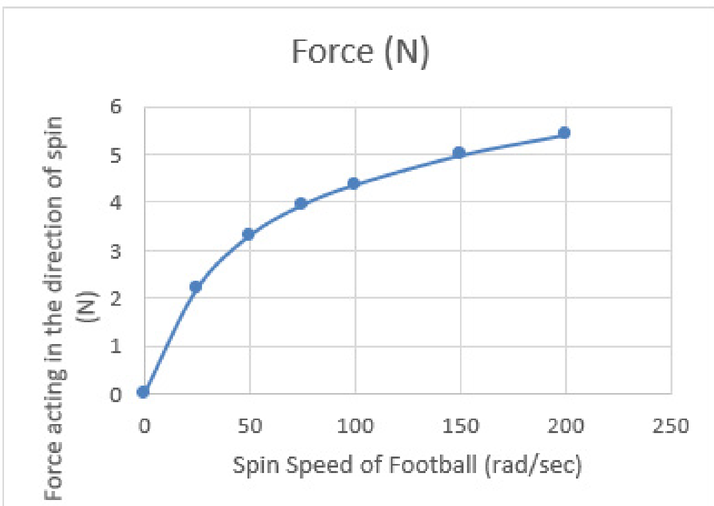

Figure 6: The plot of Lateral force vs Spin speed.

Another interesting observation from the simulation is the boundary layer separation on either side of the ball on the Z-plane. On the side where the direction of the boundary layer and air is opposite, there is a premature separation of the boundary layer while on the other side, the Boundary layer seems to be attached for a bit longer before detachment to turbulence as described by Mehta and Pallis (R. D. Mehta, 2001). This asymmetric boundary layer separation was also successfully captured by Ansys Fluent (Figure 7). Once the air reaches the maximum point, the pressure starts increasing (adverse pressure gradient) leading the velocity to decrease. Once the boundary layer separation occurs, the adverse pressure gradient is so large that it will act against the flow leading to recirculation. This region behind the ball, where there is recirculating flow is called the turbulent wake (Figure 8, Real External Flows) .

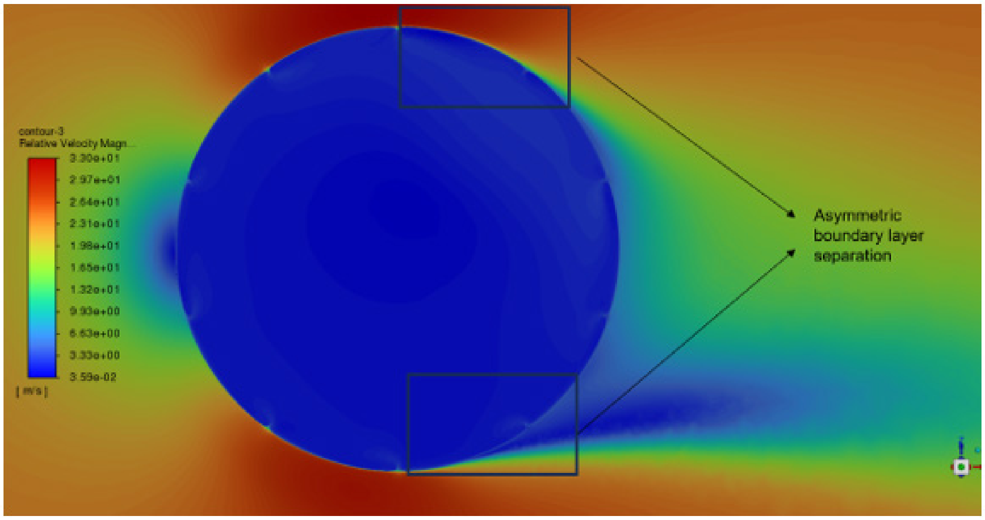

Figure 7: Figure depicting the asymmetric boundary layer separation.

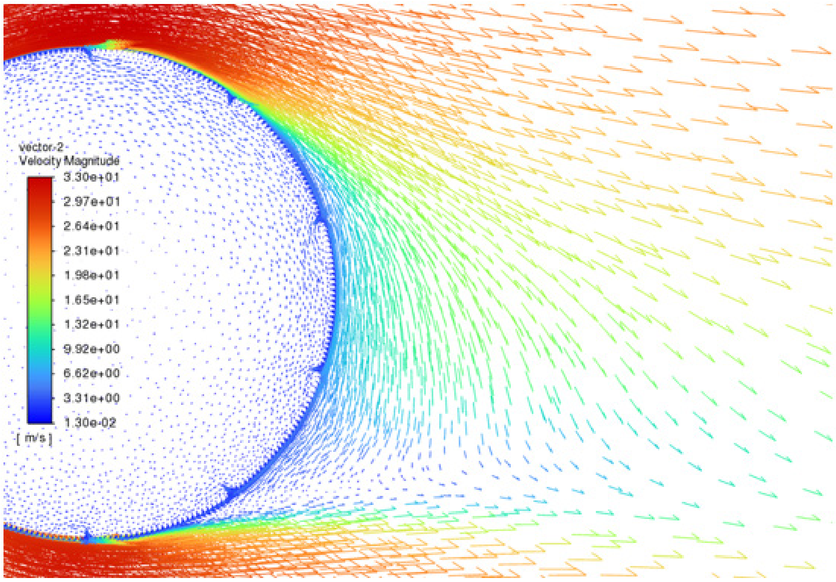

Figure 8: Turbulent wake

6. In the real world

In real life, many factors affect the movement of the football during the freekick. These include the trajectory of motion of the ball, the effect of viscous drag leading to slowing down of the ball and material as well as the structure of the football. Instead of having an icosahedron shape, the modern balls consisted of spherically molded panels of eight or six. These leads to having smoother surfaces which resulted in unpredictable movement of the ball in the air. The real issue, according to NASA (Marlaire, 2010), is the issue of the drag of the ball drastically reduces leading to uncontrollable movement of the ball.

7. Further Steps

In this case study, a simple side spinning ball was evaluated to understand the effect of the spin on the ball. This case study only checked the effect of side spin on the ball and a more detailed analysis could be carried out to understand how the structure of the ball affects movement of the ball in air. Another area of analysis which could be interesting would be to know the effect of gravity and how it affects the forces acting on the ball. The case study also assumes that the axis of rotation is constant during the flight of the ball. However, in reality, it shifts as the ball moves in the air leading to different behavior of the ball during flight. This can be another avenue of research in the future.

8. References

Briggs, L. J. (1959). Effect of spin and speed on the lateral deflection (curve) of a baseball; and the Magnus effect for smooth spheres. American Journal of Physics 27, 589-596.

Marlaire, R. D. (2010, June 18). Retrieved from NASA: https://www.nasa.gov/topics/nasalife/features/soccer_ball.html

R. D. Mehta, J. M. (2001). The aerodynamics of a tennis ball. Sports Engineering, 177-189.

Use and Reproduction

The content used in this resource may only be used or reproduced for teaching purposes; and any commercial use is strictly prohibited.

Document Information

This case study is part of a set of teaching resources to help introduce students to topics related to fluids.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com