Modeling Conventional Swing of a Cricket Ball

R. Latchman[1], A. Pooransingh[1]

[1]University of the West Indies - St. Augustine, St. Augustine, Trinidad and Tobago

Abstract

Conventional swing bowling in the sport of cricket (Figure 1) is one aerodynamic phenomenon which a bowler uses to gain an advantage over a batsman. (Baker 2009, 1949) outlines the many experimental studies done on conventional swing. Numerical analysis has, however, been limited. Using the CFD Module of COMSOL Multiphysics® software in the analysis of conventional swing will considerably reduce lead times and costs, provide the capability for solving a wider range of complex flow problems and provide comprehensive and visual information when compared to an experimental-based approach (Jiyuan, Yeoh and Liu 2008, 4 - 5). The results will be validated using the experimental results of (Mehta 2008).

A computational model of a cricket ball and the computational domain were built using the Geometry node of the Model Builder in the CFD Module (Figure 2). A multi-stage modeling strategy was implemented. For 2-D and 3-D simulations, triangular cells and tetrahedral cells were used respectively for mesh generation. Very coarse meshes were used as the starting point and were refined to obtain the most accurate solution within limitations of the computational demand and calculation turnover time requirements.

The driving force of conventional swing is based on the principle that as the ball travels through air there is turbulent flow on one side of the ball while there is laminar flow on the other side (Figure 1). A turbulent flow interface was therefore used to model the flow around the cricket ball. The Turbulent Flow, k-epsilon model was used to simulate the flow for a non-rotating ball while the Rotating Machinery, Turbulent Flow, k-epsilon user interface was used for simulating flows in which backspin of the ball was considered.

The variation in the side and drag forces will be investigated by varying the ball velocity, seam angle and backspin of the ball. The fluid flow profiles (Figures 3 and 4) correspond to the theory suggested previously (Figure 1). As the air flows around the surface of the ball, its velocity increases due to compression. A larger region of higher velocity on the seam side of the ball confirms that the air on the seam side is "tripped" into turbulence. As a result, the pressure of the fluid on the seam side is greater than on the non-seam side. This verifies that the resultant side force on the ball is toward the seam side and the ball will swing towards the seam side as expected.

Preliminary results indicate that the mechanism of conventional swing is observed, however, the turbulence model parameters need to be redefined for correlation to the experimental results of (Mehta 2008).

Using the CFD Module of COMSOL Multiphysics in the analysis of conventional swing will reduce cost while providing the capability of studying a wider range of problems. It will allow the cricket community better understand the phenomenon of conventional swing and be an integral part in the development of a conventional swing training aid.

Reference

C J. Baker, “A calculation of cricket ball trajectories” Proc. IMechE Vol. 224 Part C: J. Mechanical Engineering Science: 1947 – 1958. (2009).

R. M. P. S. Bandara and N. S. Rathnayaka. "Modelling of Conventional Swing of the Cricket Ball using Computational Fluid Dynamics." KDU International Symposium 2012, 447- 457. (2012).

Rabindra Mehta. "Sports ball aerodynamics." USA: Springer Vienna. (2008).

Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu. "Computational Fluid Dynamics: A Practical Approach." Oxford: Elsevier. (2008).

Figures used in the abstract

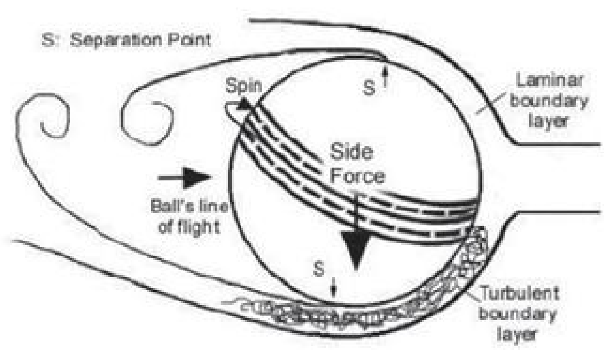

Figure 1: Schematic of flow over a cricket ball for conventional swing (Mehta 2008)

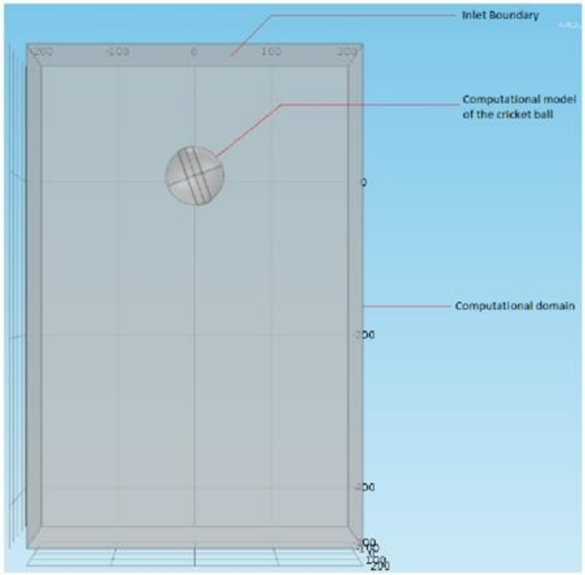

Figure 2: Computational model of the cricket ball positioned at 150 mm from the inlet boundary of the computational domain

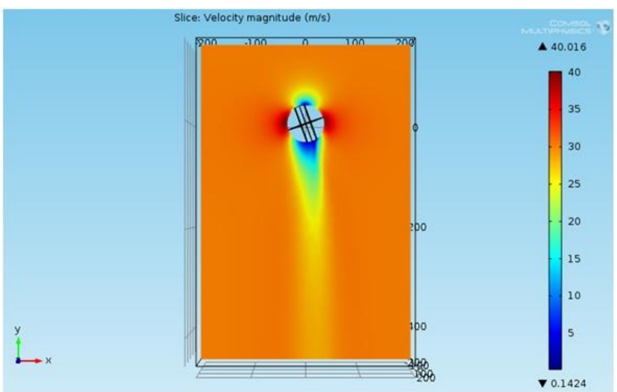

Figure 3: Velocity magnitude plot for a 3D simulation of a non-rotating ball travelling at 67 mph with a seam angle of 20°

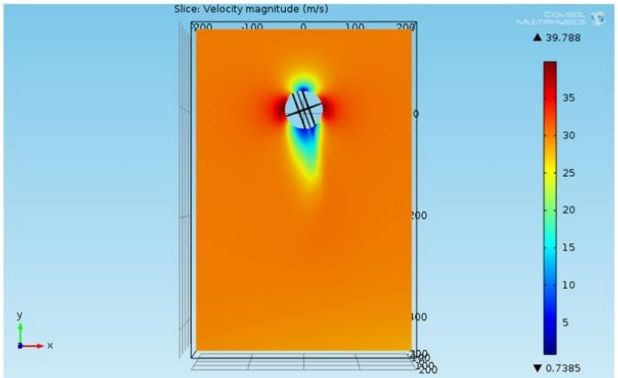

Figure 4: Velocity magnitude plot for a 3D simulation of a ball rotating at 11.4 rev/s, travelling at 67 mph with a seam angle of 20°