Simulation and understanding of the aerodynamic characteristics of a badminton shuttle

Dr John Hart*

CSER, Sheffield Hallam University, Sheffield, S10 2BP, UK

Abstract

This paper presents a study in the simulation and understanding of the aerodynamic characteristics of a badminton shuttle, using Computational Fluid Dynamics. Modeled shuttle geometry was based upon a high quality synthetic shuttle, a Yonex Mavis 370. The study investigated the use of Reynolds Averaged Navier Stokes (RANS) simulation in comparison to Scale Resolving Simulation (SRS), for the prediction of the complicated flow fields that are associated with the bluff body aerodynamics of badminton shuttles. RANS are known to struggle with predicting large bluff body separations, in comparison to SRS modeling. However SRS modeling can require significant computational resource, and the models require careful application with consideration made to spatial and temporal resolution. It was found that unsteady RANS was capable of predicting comparable time averaged flow field data to the SRS models. Both approaches produced feasible drag coefficient values for the shuttle. However RANS was incapable of predicting the complicated turbulent vortex structures within the shuttle wake, which were captured by SRS.

* Dr John Hart. Tel.: +44 (0)114 225 4405; fax: +44 (0)114 225 4356
E-mail address: john.hart@shu.ac.uk

Keywords: badminton; shuttle; computational fluid dynamics; aerodynamics
1. Introduction

Amongst sports projectiles, the badminton shuttle is incomparable in form and mass. This results in unique aerodynamic characteristics, and flight behavior of the shuttle. The aerodynamics of shuttle flight has interested researchers for a number of years. Wind tunnel and experimental investigations have been published by different research teams. The extensive work of Cooke (1992), and latterly the experimental works of Hasegawa (2013), Alam (2010) and Chan (2012) have all brought further understanding to this subject. Besides obtaining aerodynamic coefficients, Cooke (1992) conducted smoke visualization of shuttle flow structure, whilst Hasegawa (2013) used smoke flow and PIV.

The analysis of shuttle aerodynamics and associated flow behavior using Computational Fluid Dynamics (CFD) has however been extremely limited. Frank (2000) published the first CFD simulation of a shuttle model, a Reynolds Averaged Navier Stokes (RANS) solution of a simplified synthetic Carlton C-100 shuttle. Available computational resource restricted modeled geometry and computational mesh to 10,000 tetrahedral surface elements with volume cell count in the region of 700,000. This was a large simulation for the time though small by current standards. Frank reported an over prediction in drag of 19% compared to experiment, Cooke (1992). More recently Verma (2013) published a study comparing aerodynamics of a synthetic Yonex Mavis 350, a Yonex feather shuttle, and a generic solid skirt model. Verma chose a computational mesh of 3.2 million elements to simulate the synthetic shuttle, based on a mesh dependency study which only compared two different meshes. Simulation was based on steady RANS, with results claimed in good agreement with experiment.

In the recent review of sports projectile aerodynamics, Goff (2013) stated that what was needed in the aerodynamic study of shuttles was CFD analysis. However Goff cautioned that this would be challenging due to the geometry of a shuttle. Indeed the accurate recreation of the complex flow physics and phenomena, attributable to the geometry of these projectiles, poses a considerable simulation challenge.

Published CFD studies to date have been RANS solutions only. Although these provide a good starting point for understanding the flow field around a shuttle, they do not fully reveal the complex underlying flow phenomena. Boundary layers and separation can be predicted by a well tuned RANS simulation, but the prediction of large separation regions is problematic, Spalart (2009). RANS therefore perform best when simulating wall bounded flows and can struggle with bluff bodies and shear layers. The majority of sports projectiles, including shuttles can be classed as bluff bodies. Scale Resolving Simulation (SRS) models typically produce better results in such flow cases, resolving flow details that cannot be obtained through RANS. The use of SRS models has potential therefore to provide detailed understanding of shuttle aerodynamics. SRS models are computationally expensive compared to RANS and have to be applied carefully.

This paper presents the first stages of a CFD investigation of shuttle aerodynamics, presenting results for a synthetic shuttle. The paper discusses the application of CFD to the study of the synthetic shuttle, and results of applying SRS models to the problem.

2. Problem Definition

Modeled shuttle geometry was based upon a Yonex Mavis 370, a high quality synthetic shuttle. Shuttle geometry was constructed in PTC Pro/ENGINEER Wildfire 5.0. All measurements were made using digital verniers, and a digital vernier height gauge on a granite measurement slab. Repeat measurements were taken radially around the skirt of small scale features due to the fineness of detail. For example the finest features at the base of the skirt, which are semi-circular in cross-section, had an average measured depth of 0.15 mm and width of 0.4 mm. Modeled detail is shown in figure 1. The shuttle was determined to have a height H = 78.6 mm and a maximum skirt diameter D = 66 mm. The maximum skirt diameter D is of importance as it is accepted practice to use this length scale when calculating the frontal area for use in determining the coefficient of drag of shuttles.
2.1. Boundary Conditions

The computational domain was cylindrical in shape, with the cylindrical domain and shuttle axis aligned. The flow inlet (velocity inlet) was placed 5H upstream of the shuttle, and the flow outlet (pressure outlet) 15H downstream of the shuttle. The diameter of the cylindrical domain was 9D, with outer walls modeled as slip walls. An inlet flow velocity of 20 m/s (Re = 90365) was used for all simulations, unless otherwise specified. The shuttle was modeled as non rotating. Turbulence values at flow boundaries were specified using turbulence intensity \( T_i = 1\% \) and a turbulent viscosity ratio \( TVR = 2 \).

2.2. Validation

Validation data was provided by a short wind tunnel investigation using a TecQuipment AF100 wind tunnel. The tunnel is an open circuit tunnel, with square working section 305 mm x 305 mm. The tunnel has a maximum speed of 36 m/s. Drag coefficient was measured for a Yonex Mavis 370 shuttle centrally sting mounted in the working section, in 5 m/s increments from 15 m/s to 35 m/s. Reported values are based on time averaged repeat measurements of 30 second duration data samples. Tunnel \( T_i \) value is unknown, validation data should at best be considered qualitative, however good agreement was found with previous shuttle studies, Alam (2010).

3. Computational setup

All simulations were conducted using the commercial CFD code ANSYS-Fluent V14. Initial mesh dependency simulations were conducted using steady state RANS with turbulence closure provided by the k-\( \omega \) SST model sensitized for low Re affects. This is a well respected RANS model for the simulation of separated flows. Spatial discretisation of the momentum and turbulence equations was second order accuracy. A SIMPLE pressure interpolation scheme provided pressure velocity coupling. The use of Scale Resolving Simulation (SRS) turbulence models was also investigated using models of increasing complexity; SAS-SST (Scale Adaptive Simulation SST), DDES-SST (Delayed Detached Eddy Simulation SST), and LES (Large Eddy Scale - WALE). LES modeling attempts to resolve the largest scales of turbulence and model the smallest scales. However the computational effort and resource required, to resolve even the largest scales in near wall regions, is great. This generally restricts the use of the model to free stream flows and research. SAS-SST and DDES-SST are essentially hybrid turbulence models which combine RANS simulation in near wall regions, and attempt to resolve the large scales away from walls (LES). This reduces required computational effort. All SRS simulations in this study used second order spatial and temporal discretisation for all equations, with the exception of the momentum equations. These were discretised using a bounded central differencing scheme, as second order schemes are numerically diffusive for SRS models. An extensive discussion of SRS numerical schemes and application of SRS models will not be recounted here and can be found in Menter (2012).

All simulations were performed on a SGI Altix HPC using parallel distribution over 32 processing cores. Simulations were monitored for residual, force coefficient, and variable convergence.
3.1. Mesh

Several computational meshes were constructed to study the sensitivity of numerical solution to mesh resolution, Table 1. All mesh were 3D, and hybrid in structure. Tetrahedral surface mesh for the shuttle and flow domain boundaries were constructed using ANSYS-Meshing. Three different surface mesh resolutions (Mesh A-C) were created for the shuttle, ranging from 210,000 elements (minimum element size 0.3 mm) up to 1.1 million elements (minimum element size 0.075 mm). Surface resolution for the smallest geometric features was representative for the coarsest surface mesh, whilst the finest surface mesh accurately captured the surface curvature of all skirt details. These meshes were exported to ANSYS-TGrid 14.5 for construction of the volume mesh. The main body of all volume mesh was constructed using HexCore, a hanging node hexahedral cell structure. Mesh C was systematically refined to create Mesh D, E, and F. Five layers of prismatic cells at wall boundaries were first added to improve the resolution of surface boundary layers and viscous sub-layer in simulation. The addition of prismatic layers reduced wall $y^+$ values, the normalized distance to the wall surface from the centre of the first computational cell. This value should ideally be close to $y^+ = 1$. Further systematic refinements to the HexCore mesh through and behind the skirt were then made to improve resolution of wake structures, and associated flow phenomena. Constructed mesh size ranged from 1.84 million cells, to a maximum of 21.19 million cells.

Table 1. Computational mesh

| Mesh | Surface resolution (million elements) | Prismatic layers | Wake refinement | Volume cell count (millions) | $y^+$ min. | $y^+$ max. | $y^+$ avg. | $C_D$ |
|------|--------------------------------------|-----------------|----------------|-------------------------------|------------|------------|-----------|------|
| A    | Coarse (0.21)                         | No              | No             | 1.84                          | 0.28       | 30.35      | 7.54      | 0.67 |
| B    | Medium (0.63)                         | No              | No             | 5.20                          | 0.48       | 17.63      | 4.87      | 0.65 |
| C    | Fine (1.1)                            | No              | No             | 8.25                          | 0.13       | 12.55      | 3.82      | 0.65 |
| D    | Fine (1.1)                            | Yes             | No             | 14.11                         | 0.06       | 2.62       | 1.11      | 0.65 |
| E    | Fine (1.1)                            | Yes             | Yes            | 18.51                         | 0.07       | 2.57       | 1.07      | 0.60 |
| F    | Fine (1.1)                            | Yes             | Yes            | 21.19                         | 0.10       | 2.55       | 1.07      | 0.61 |

4. Steady RANS simulation

Steady RANS simulation was run for each computational mesh at $Re = 90365$. Predicted $C_D$ and wall mesh $y^+$ values are shown in Table 1. It can be seen that it was only through use of prismatic mesh layers that a near ideal $y^+$ value was achieved. However wake refinement was required to achieve a mesh independent result, with predicted $C_D \approx 0.6$. Figure 2 clearly reveals the influence of the mesh on the predicted flow structures through the shuttle. It is only once the wake has been properly refined that the large vortex core behind the shuttle base forms,
as do the circulating flow features in the shuttle skirt, and the significant air jets entering the skirt at its mid section. Although the simulations were run as steady, it is clearly possible to see unsteady behavior beginning to appear in Mesh F attributable to the mesh refinement. Mesh F was chosen for all further simulation. A series of simulations were also conducted varying Re = 45000 to Re = 270000 (10 - 60 m/s), the results of which are compared to values obtained from the wind tunnel study. These are shown to be in reasonable agreement, figure 3. Wind tunnel C_D decreases with increasing Re above Re = 110000 which it is hypothesized is attributable to skirt deformation, at higher speeds, an artifact that is not recreated in the CFD.

5. Scale resolving simulation

All SRS studies were conducted on Mesh F at Re = 90365. Steady RANS simulation was switched to unsteady (URANS) time dependent mode and run until statistically settled. This simulation was then used as the starting point for all SRS models. A time step T_s = 2.5x10^{-4} s was applied for all simulations. URANS k-ω SST predicted a slightly higher drag coefficient C_D = 0.62 than RANS. SRS predicted lower drag coefficient SAS-SST C_D = 0.57, DDES-SST C_D = 0.57, LES C_D = 0.55. A comparison of predicted shuttle wake structure is shown in figure 4. This shows iso surface of Q-Criterion, a variable that reveals core locations of turbulent vortices and structures, and avoids the display of shear layers that may conceal structure, Menter (2012). It is clear that URANS only predicts large scale structure and does not predict any of the small scale structures in the central core of the wake behind the skirt. Comparing time averaged velocity flow fields predicted by URANS and SRS it can be seen that URANS is capable of producing comparable averaged features, figure 5. This shows URANS can provide an overview of flow structure. However SRS models are required to understand the full turbulent structure. The hybrid SRS models were found to give comparable performance to full LES. SRS models are however affected significantly by mesh quality, predicted structures would become finer if Mesh F was further refined, however this would not benefit URANS.

Fig. 3. Comparison of RANS predicted drag coefficient with wind tunnel

Fig. 4. Comparison of Q-Criterion Q = 10^5 structures coloured by velocity
Structure is also influenced by time step size, as shown in figure 6, and must therefore be carefully applied. Reducing the time step by an order of magnitude increased the detail of resolved structure. Predicted drag coefficient reduced from LES $C_D = 0.55$ at $T_s = 2.5 \times 10^{-4}$ s to LES $C_D = 0.54$ at $T_s = 2.5 \times 10^{-5}$ s.

6. Conclusions

A study of the CFD modeling of a synthetic Yonex Mavis 370 shuttle has been conducted. This investigated the use of RANS and SRS in modeling the complex flow phenomena associated with these bluff body projectiles. RANS simulations were shown capable of predicting time averaged flow phenomena, as resolved by SRS models. However RANS was not capable of resolving the complex time dependent turbulent flow structures revealed by SRS. Further investigation of SRS modeling needs to be conducted including further mesh refinement.

References

Alam, F., Chowdhury, H., Theppadungporn, C., Subic, A., 2010. Measurements of aerodynamic properties of badminton shuttlecocks. In: Procedia Engineering 2, Proceedings of 8th International Conference of the International Sports Engineering Association, pp. 2487-2492
Chan, C.M., Rossmann, J.S., 2012. Badminton shuttlecock aerodynamics: synthesizing experiment and theory, Sports Engineering 15:61-71
Cooke, A.J., 1992. The Aerodynamics and Mechanics of Shuttlecocks. PhD thesis, Cambridge University, U.K.
Frank, M.H., Cooke, A.J., Cant, R.S., 2000. Computational Fluid Dynamics of a Shuttlecock. In: Proceedings of Third International Conference on the Engineering of Sport, Sydney, Australia, June 10-12, pp. 371-376
Goff, J.E., 2013. A review of recent research into aerodynamics of sport projectiles. Sports Engineering 16:137-154
Hasegawa, H., Kitta, S., Murakami, M., 2013. Flow analysis and aerodynamic characteristics of a badminton shuttlecock with spin at high Reynolds numbers. Sports Engineering 16:91-98
Menter, F.R., 2012. Best Practice: Scale-Resolving Simulations in ANSYS CFD, Version 1.0. Last accessed on 31/10/13 at URL:http://www.ansys.com/Resource+Library/Technical+Briefs/Best+Practice+%3E+Scale-Resolving+Simulations+%3E+ANSYS+CFD
Spalart, P.R., 2009. Detached-Eddy Simulation. Annu. Rev. Fluid Mech. 41:181-202
Verma, A., Desai, A., Mittal, S., 2013. Aerodynamics of badminton shuttlecocks. Journal of Fluids and Structures 41:89-98