Design of Stadia for Hot Climates: A CFD and Wind Tunnel Analysis of the Wind Flow over a Saddle-Shaped Roof

Rubina Ramponi[1,2], Paul Lynch[1], Giulia Matteoni[1], Mutlu Ucuncu[1], Steve Walker[1]
[1]Advanced Technology & Research, Arup, London, UK
[2]Advanced Technology & Research, Arup, San Francisco, USA

Abstract

The roof shape and orientation are crucial factors in the design of stadia for very hot climates, to regulate the wind and solar access to the bowl and ensure usability.

Computational Fluid Dynamics (CFD) has been used to model the airflow in and around stadia and interface with other modelling tools to predict thermal comfort. However, relatively few experimental studies provide validation data to support simulations of geometrically complex structures.

This paper presents a CFD and experimental study of a stadium with double saddle-shaped open roof. CFD results are compared with measurements carried out with Particle Image Velocimetry. The performance of the concave and convex roof profiles are discussed.

Introduction

The increasing number of large international sporting events hosted in hot climates pose new challenges to engineers and designers. Design aspirations to retain the outdoor character of sports such as athletics and soccer lead to stadia with a large roof oculus. This conflicts, however, with the need to protect the bowl from the outdoor conditions that are very hot, and mechanical cooling is likely to be needed during the hotter seasons to meet heat stress and thermal comfort limits. It follows that there is a central design challenge of preventing hot outside air mixing with the cooler air supplied into the competitor and spectator areas (Ucuncu et al. 2010).

Thermal comfort is influenced by several environmental variables, including air temperature, relative humidity, mean radiant temperature and air speed. The shape of the stadium affects the wind and solar penetration to the field of play, track (if present) and the tiers and, therefore, has a direct impact on thermal comfort and energy consumption for active cooling. In particular, a key design factor is the shape and orientation of the roof and its oculus. In very hot climates, where the external air temperature tends to be higher than the comfort threshold, it is vital to limit external hot air from penetrating into the stadium and design feedback is crucial at the early stages to guide the design towards feasible solutions.

Past studies have analysed the relationship between stadium design and airflow conditions inside a stadium, using either wind tunnel testing or Computational Fluid Dynamics (CFD). Among others, Szucs et al. (2009) carried out a parametric analysis of the impact of the roof geometry and façade porosity on the ventilation of a generic stadium by means of wind tunnel testing. Bouyer et al. (2007) used wind tunnel measurements and an in-house tool to assess the thermal comfort inside two stadia with different geometries under the same climate. However, their study considered uniform temperature and humidity conditions inside the stadia and therefore does not provide a spatial distribution of the comfort levels. Van Hooff and Blocken (2011) built upon the work done by Persoon et al. (2008) on 2D generic stadium configurations and performed a parametric study of the effect of the roof arrangement on the internal wind flow and wind driven rain within different stadia using 3D numerical simulations.

CFD simulations are used to predict the airflow in and around stadia. CFD overcomes the limitations of wind tunnel testing in modelling the thermal buoyancy effects, provides resolution of every point of the domain and gives the opportunity to interface with other modelling tools to perform further calculations such as thermal comfort or exposure to solar radiation. Comparison with wind tunnel data is recommended to support the reliability of CFD results. However, relatively few validated CFD studies (e.g. van Hooff and Blocken, 2010a, b, Blocken and Persoon, 2009) can be found in the literature for geometrically complex and physically large structures.

One of the barriers to validating the results of CFD simulations of buildings with unusual shapes is the lack of availability of high-resolution experimental data. In this paper, we provide results obtained with Particle Image Velocimetry (PIV), an optical technique that captures the instantaneous velocity fields at high spatial resolution over an extended 2D region, and therefore allows detailed flow comparisons with CFD (Ramponi et al. 2017).

This paper presents a study of the airflow in and around a double-roofed stadium with a saddle-shaped open roof. An overview of the methodology used to simulate and evaluate the stadium cooling performance is first presented to highlight the role played by the airflow simulations in the comfort assessments. Results of CFD simulations for one roof profile are then compared with PIV measurements made in a boundary layer wind tunnel. The effect of the roof profile on the expected cooling performance of the open roof stadium is evaluated based on the amount of conditioned air lost through the stadium roof.

Stadium geometry

The tests were performed on a stadium featuring a double-skinned roof with a saddle shape. The roof serves several purposes, not least an architectural one. From the perspective of thermal comfort, it is an effective solar shade despite its very lightweight design since shading from radiation is important in hot conditions. The ventilated cavity that is formed by the double-skin generates a wind flow path, and is intended to minimise wind driven mixing of hot outside air with the volume it encloses.

Figure 1 shows the saddle geometry of the roof that creates a convex and concave profile depending on the wind direction. The roof cavity at the oculus is twice as deep as along the perimeter and is ventilated through a 50% porous mesh on both ends. The stadium is modelled as isolated without nearby buildings or topographic features due to uncertainty in the massing and build schedule of these buildings.

The studies presented here focus on the flow aligned with the convex and concave profiles. These are expected to bound the cooling performance of the intermediate angles.

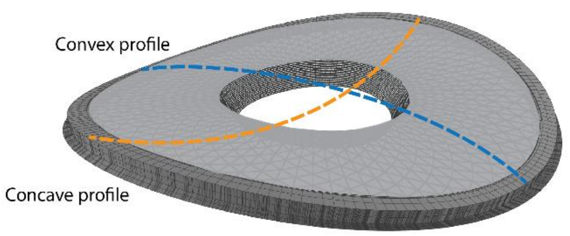

Figure 1: Saddle-shaped stadium roof.

Methodology

Overview of the cooling performance assessment

The cooling performance of the stadium was assessed through a combination of different numerical simulations and the results were used to calculate the amount of conditioned air that would be needed to provide suitable thermal comfort conditions and prevent heat stress.

A comprehensive assessment of the environmental conditions inside the stadium requires the simultaneous solution of several variables, including air temperature, wind speed and mean radiant temperature at every point of the area of interest at different times of the year. This approach would be very onerous and infeasible to tackle in a single CFD model. Therefore, the simulation of the airflow and thermal radiant field were decoupled as shown in Figure 2 and thermal comfort and heat stress were evaluated with in-house developed scripts using the computational software Mathematica (Wolfram Inc., 2016). CFD simulations provided air speed, temperature and humidity data at a high-spatial resolution and therefore facilitated close integration with other modelling tools.

The long-wave component of the Mean Radiant Temperature (MRT) was calculated based on the surface temperatures obtained from Dynamic Thermal Modelling (DTM) analysis carried out in the Arup in-house developed DTM tool Oasys Ltd. ROOM (Holmes 1991) with some modifications required to model stadia (White 2009). The short-wave component was obtained from a solar analysis using Radiance (Ward, 1994).

This paper focuses on the CFD modelling aspects of this process. Given the importance of accurately modelling the interaction between the wind and internal environment for the temperature and comfort distribution, a study was carried out to compare the results of the CFD model to experimental results. The outcomes of this study are presented in the following sections.

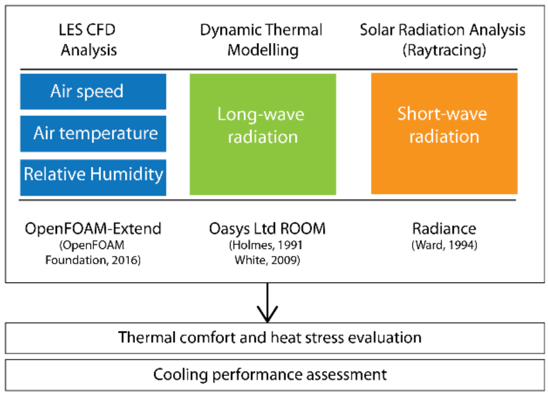

Figure 2: Summary of the assessment methodology

Particle Image Velocimetry measurements

PIV measurements were carried out at the boundary layer wind tunnel of BMT Fluid Mechanics using a 1:200 scale model of the proposed stadium (tunnel dimensions: 4.8m wide x 2.4m high x 15m long). The test model was mounted on a turning table that allowed testing of multiple wind directions.

The PIV system consisted of a low-speed laser from Litron, capable of laser pulse pairs with a repetition rate of up to 15Hz with output energy of 200mJ at 532nm, a Bobcat FlowSenseEO 16M camera with resolution of 16 MegaPixels and a variable aperture 135mm lens. The laser unit was mounted on the tunnel roof, with the laser sheet illuminating a vertical plane across the roof cavity, as illustrated in Figure 3. To minimise effects of light reflection, the Stadium model was painted black. The test chamber was seeded with a uniform concentration of PolyethyleneGlycol (PEG) mixed at a volume ratio of 4 parts warm water to one part PEG. The measurements were performed for the directions corresponding to the concave and convex roof profiles at a reference wind speed of 1.7m/s as measured by a Pitot-static probe at a full-scale height of 95m. The data were acquired in 13 batches of 80 image pairs at a frequency of 2Hz at full camera resolution. An example of a flow visualisation image from the PIV measurements is shown in Figure 4.

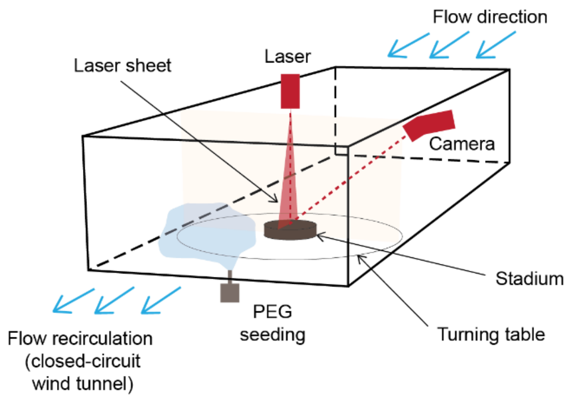

Figure 3 PIV setup in the wind tunnel (Ramponi et al. 2017)

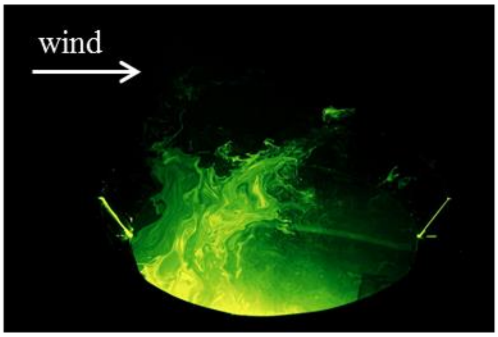

Figure 4: Snapshot of the PIV measurements of the concave profile.

Computational Fluid Dynamics simulations

The CFD analysis was undertaken using OpenFOAM-Extend 1.6. The computational grid consisted of approximately 15 million predominantly hexahedral elements.

The domain size (2000m streamwise, 1600m wide, 310m tall) was selected based on the recommendations of Tominaga et al. (2008), with a blockage ratio of approximately 3%.

The discretised, incompressible Navier-Stokes flow equations were solved using the PISO algorithm (Issa 1986) for unsteady, incompressible flow.

Turbulence was modelled using a Large Eddy Simulation approach, with a one-equation sub-grid scale eddy viscosity model (De Villiers 2006). Second order numerical schemes were used for the time derivatives and spatial derivatives of pressure and momentum.

Synthetic turbulence was introduced at the inlet to the computational domain, using the LEMOS Turbulent Inlet Generator by Kornev et al. (2008). Profiles of mean velocity, streamwise integral length scale and Reynolds stress were defined at the inlet, based on a site terrain analysis, following the methodology in ESDU 01008 (2010).

The side and top boundaries of the domain were modelled as free-slip boundaries. The downstream boundary was prescribed a hydrostatic pressure distribution. The ground and stadium surfaces were prescribed as adiabatic no-slip boundaries. Heat transfer between solid surfaces and the air was not modelled, as it was expected to be of secondary importance to the heat transport due to wind scouring.

The largest mesh cell size was selected to ensure the turbulent statistics were maintained throughout the domain. Close to the stadium, the cell size was reduced to approximately 0.5m to capture the separation and mixing in and around the roof and stadium bowl, with further refinement for the cells at solid boundaries.

The mean wind speed profile was approximately equivalent to a power-law profile with reference wind speed of 11.3m/s at 10m height and an exponent of 0.22.

Each run consisted of 15 minutes of simulated time, with the first 5 minutes discarded to allow the synthetic turbulence within the domain to approach a quasi-steady condition.

For comparison with the wind tunnel measurements, the CFD analyses presented herein were carried out without solving for buoyancy. However, due to the importance of the negative buoyancy of the cool air supplied to the stadium bowl, the simulations used for the stadium design incorporated a Boussinesq buoyancy model.

Results and discussion

Overall flow characteristics

A simplified representation of the flow patterns around the stadium for the concave and convex roof profiles is presented in Figure 5. The wind flowing through the convex profile stays attached to the roof. The roof cavity generates a shear layer that allows only a very weak flow to penetrate to the bowl, leaving the internal volume relatively isolated. The concave profile induces a stronger flow separation that creates a large and unstable separation bubble. The unsteady collapse of the bubble leads to a non-stationary behaviour of the flow in the roof oculus and a stronger mixing between the wind and the internal air.

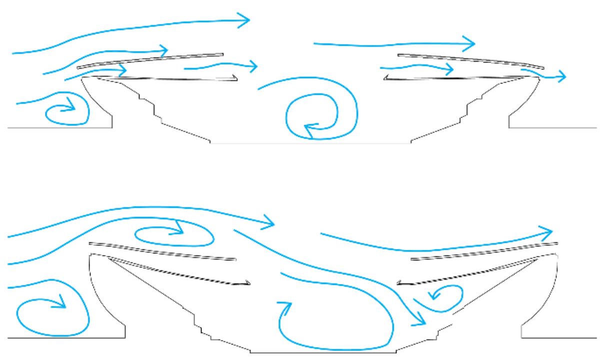

Figure 5: Simplified flow patterns in and around the stadium for the convex (above) and concave (below) roof profiles

Comparison between PIV and CFD

The numerical results obtained for the concave roof profile were compared with the experiments. Figure 6 shows a comparison between the streamwise relative velocity (U_x/U_{ref}) with respect to the freestream velocity U_{ref} measured at 95m above ground with PIV and simulated with CFD along different vertical lines across the oculus: V2 (upstream), V5 (centreline) and V8 (downstream). The blue area indicates the mean wind speed variation across the recorded batches of PIV measurements. The grey area marks the total thickness of the double-roof (H).

It is noticeable that at the upper roof level (y/H=0), the wind speed tends to increase going from the upstream (V2) to the downstream (V8) line across the oculus and ranges from 0.2 to 0.4. In all positions, the wind speed decreases gradually with height.

The CFD results are mostly within the PIV velocity bands along V2 and V5, and are slightly higher than the PIV within the roof cavity along the downstream line V8. The differences in the average streamwise velocity profile obtained with CFD and PIV correspond to a Coefficient of Variation of the Root Mean Square Error (CVRMSE) as defined in ASHRAE Guideline 14-2012 (ASHRAE 2002) of 10.4% for V2, 11.7% for V5, and 12.6% for V8.

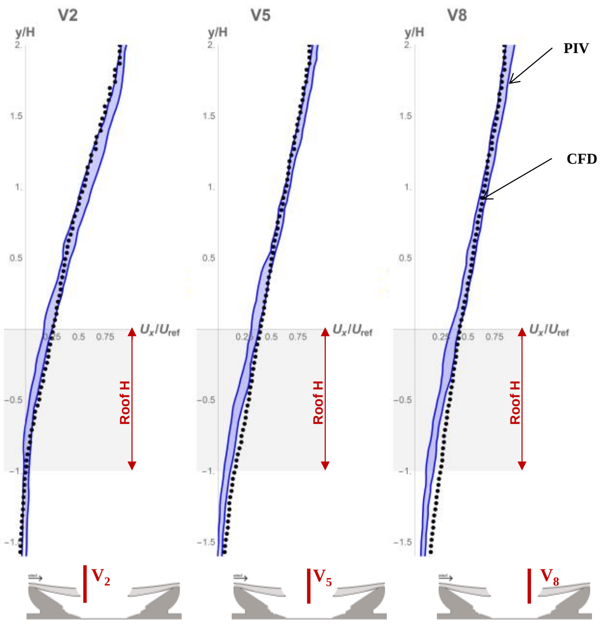

Figure 6 Streamwise velocity along three vertical lines across the oculus.

Figure 7 illustrates the variation of the vertical component of the relative vertical velocity across the oculus at the upper and lower roof level. It can be observed that towards the downstream side of the oculus (x/R>0.3, where R is the radius of the oculus) the flow creates an uplift at upper roof level (H1) and a downdraft at lower roof level (H3). The CFD results along H1 and H3 are mostly within the PIV speed bands, but underestimate the downstream uplift at upper roof level, probably due to the differing representation of the 50% porous mesh in the wind tunnel and CFD model.

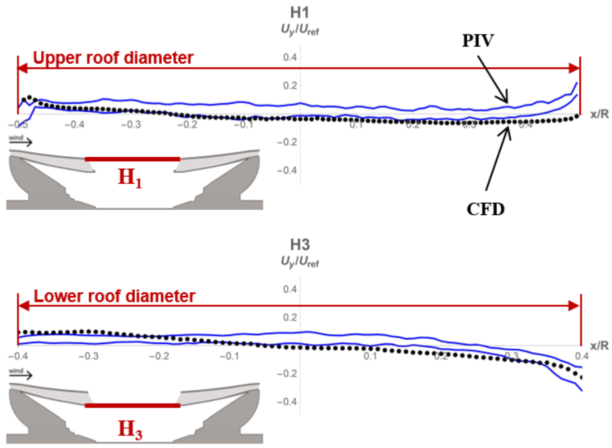

Figure 7 Vertical velocity along the horizontal lines across the upper and lower side of the oculus

Figure 8 illustrates a visual comparison between the relative velocity vector fields obtained with PIV measurements and CFD simulations in the centre plane of the stadium with the concave roof profile.

The main features of the flow are present in both datasets. Firstly, the jet of external air enters the stadium and creates a recirculation vortex in the downstream side of the bowl. Secondly, the flow separates downstream of the oculus and causes an uplift of the higher layer of the air and a deflection of the lower layer towards the tiers. As will be discussed later, the greater extents of the CFD results (Figure 9) provide insights on the flow beyond the PIV measurement area and flow separation at the roof and the upstream vortex can be observed.

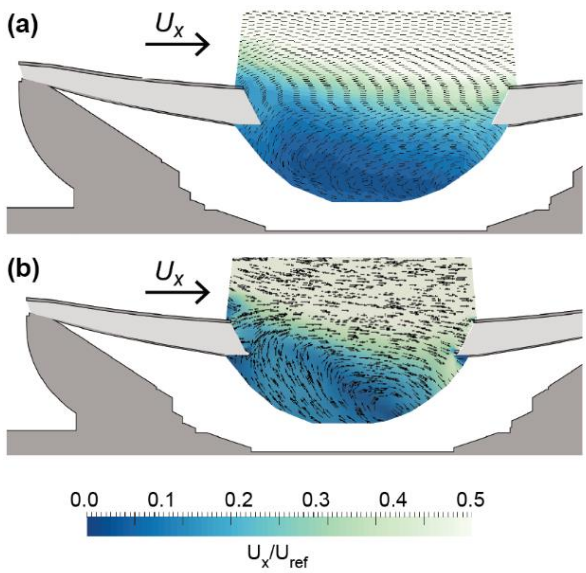

Figure 8: Relative streamwise velocity U_x/U_{ref} across the concave roof: (a) PIV measurements, (b) CFD

Performance of the stadium for the two roof profiles

Figure 9 illustrates the streamwise velocity vector fields for the convex and concave roof profiles. The concave profile is characterized by a strong separation zone at the upstream perimeter of the roof that increases the turbulence of the flow and therefore the mixing of the internal and external flow across the oculus. The flow reattaches to the roof on the downstream side of the oculus and partially deflects towards the bowl.

In contrast, the convex profile induces the creation of an attached boundary layer along the outer roof shell. Together with the ventilated cavity, this helps to create a shear layer that prevents the external flow from penetrating into the stadium.

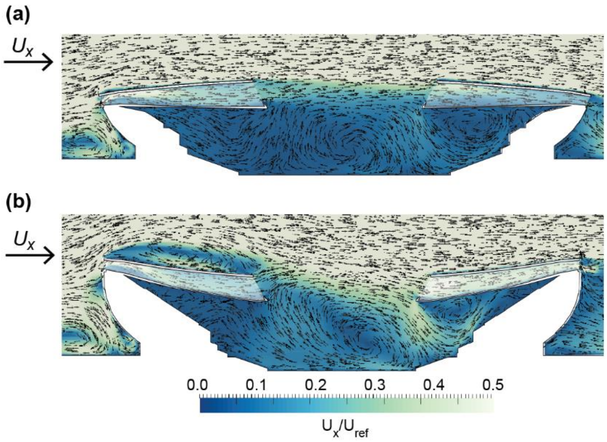

Figure 9: Relative streamwise velocities along the centreline of the stadium for different roof profiles obtained with the CFD simulations

The differences in the flow behaviour can also be noted in the average relative streamwise wind speed profiles obtained along the centreline of the oculus for the two cases and shown in Figure 10. In the case of the convex profile, the average wind speed above roof level is much higher than in the stadium, while in the case of the concave profile the average speed increases gradually with height. This is the result of the highly turbulent mixing process between the external and internal air caused by the separation at the edge of the concave roof, resulting in much greater exchange of momentum between the flows above and below the roof level.

The cooling performance of the stadium is challenging to measure with a single value. However, the rate of loss of conditioned air from the stadium bowl due to wind scour is a useful indicative proxy.

The air changes per hour presented in Table 1 represent the exfiltration of conditioned air from the stadium bowl, rather than the total air exchange across an arbitrary plane at the roof oculus, which would be much larger. This measure quantifies the impact of wind scour on the cooling system in lost cooling energy, and demonstrates that the separation from the outer edge of the concave roof profile results in greatly increased mixing and scouring by the wind in this orientation, which therefore impacts on the energy performance of the stadium. It is worth noting that the cooling system design was also dependent on other factors such as the climatic conditions associated with specific wind conditions.

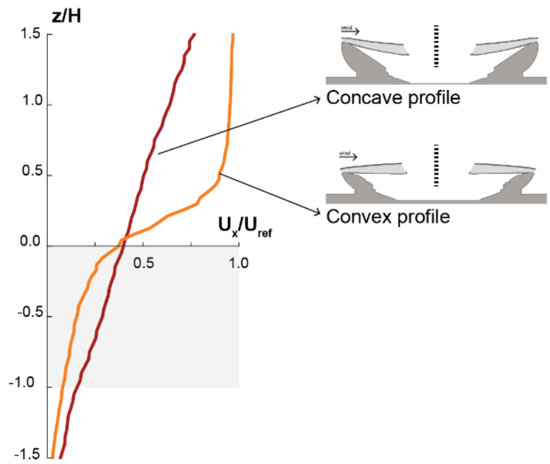

Figure 10: Relative streamwise velocities along the centreline of the stadium for different roof profiles

Table 1: Air change per hour of conditioned air from the stadium bowl

Roof profile	ACH$_{mean}$
Concave	2.9
Convex	1.8

Conclusions

The paper presents a numerical and experimental study of the impact of the saddle-shaped roof profile on the airflow around a double-roofed stadium. The study is part of a wider investigation on the relationship between roof design and thermal comfort for stadia in hot climates. Under challenging climate conditions, it is crucial to prevent the penetration of hot air inside the stadium to avoid heat stress for spectators and players.

CFD simulations have been identified as the most suitable airflow simulation tool for this situation because they provide high resolution results, capture buoyancy effects and model a mixed condition of mechanically cooled air and wind driven outside air. To support the reliability of CFD to realistically capture a complex flow, however, the numerical results (isothermal case) were compared against data acquired with PIV and other wind tunnel tests e.g. Irwin probe data (not shown in this paper).

Isothermal simulations do not account for the influence of buoyancy on the flow patterns, and so the simulations used for the design assessment did include solving for the buoyancy of the air. Unfortunately, wind tunnel methods do not allow for reliable testing of the full interaction of wind-driven momentum and thermal buoyancy, and so there remains a degree of uncertainty in the final simulations. However, validation of the isothermal flow modelling demonstrates that many complex mechanisms (e.g. flow separation, turbulent content, etc.) are well captured, and so this uncertainty is minimised.

This study considers the wind directions along the concave and convex profiles of the saddle-shape roof, as they are expected to bounding cases of conditions within the bowl. The comparison between the profiles shows a very different behaviour of the relative streamwise velocities along the vertical centreline of the roof: while the concave roof profile results in flow separation and more vigorous mixing with the internal air, the convex profile combined with the ventilated cavity promotes the creation of a shear layer that separates the external from the internal flow. Both cases are inherently challenging for CFD simulations to capture with fidelity.

It is concluded that there is a good agreement between CFD predictions and wind tunnel measurements with PIV, providing a detailed and rigorous basis for comparison with CFD. The CFD simulations not only correctly reproduce the overall flow patterns around the stadium for the most turbulent profile, but also provide relative velocity values that are in close agreement with PIV data.

Acknowledgement

The authors would like to thank Kevin Gouder of Imperial College London and BMT Fluid mechanics who conducted the experimental work.

References

ASHRAE (2002). ASHRAE Guideline 14-2002. Measurement of Energy and Demand Savings, Atlanta: ASHRAE.

Blocken, B., Persoon, J. (2009). Pedestrian wind comfort around a large football stadium in an urban environment: CFD simulation, validation and application of the new Dutch wind nuisance standard. *Journal of Wind Engineering and Industrial Aerodynamics, 97(5–6), 255–270.*

Bouyer, J., Vinet, J., Delpech, P., & Carré, S. (2007). Thermal comfort assessment in semi-outdoor environments: application to comfort study in stadia. *Journal of Wind Engineering and Industrial Aerodynamics*, *95*(9), 963-976.

De Villiers, E. (2007). *The potential of large eddy simulation for the modelling of wall bounded flows* (Doctoral dissertation, University of London).

ESDU 01008 (2010) Computer program for wind speeds and turbulence properties: flat or hilly sites in terrain with roughness changes.

Fiala, D. L. K. J., & Lomas, K. J. (1999). Application of a computer model predicting human thermal responses to the design of sports stadia. In *Chartered Institution of Building Services Engineers National Conference (CIBSE 99)*, Harrogate, UK.

Holmes, M. J., Conner, P. A. 1991. Room a computer program to predict comfort at any point in a space, *CIBSE National Conference*.

Issa, R. I., (1986). Solution of the implicitly discretised fluid flow equations by operator-splitting. *Journal of Computational Physics*, 62(1), 40-65

Kornev, N., Kröger, H., & Hassel, E. (2008). Synthesis of homogeneous anisotropic turbulent fields with prescribed second-order statistics by the random spots method. *Communications in Numerical Methods in Engineering*, *24*(10), 875-877.

Szucs, A., Moreau, S., & Allard, F. (2009). Aspects of stadium design for warm climates. *Building and Environment*, *44*(6), 1206-1214.

OpenFoam Foundation. (2016). OpenFoam user guide. Version 4.0, available at: cfd.direct/openfoam/user-guide/

Persoon, J., van Hooff, T., Blocken, B., Carmeliet, J., & de Wit, M. H. (2008). On the impact of roof geometry on rain shelter in football stadia. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(8), 1274-1293

Ramponi, R., Matteoni, G., Lynch, P., Ucuncu, M. & Walker, S. (2017). CFD and PIV study of the airflow in and around a saddle-shaped double-roofed stadium. *In Proceedings of Conference: European and African Conference of Wind Engineering (in press)*

Van Hooff, T., & Blocken, B. (2010). On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. *Computers & Fluids*, 39(7), 1146-1155.

Van Hooff, T., & Blocken, B. (2010). Coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: A case study for the Amsterdam ArenA stadium. *Environmental Modelling & Software*, *25*(1), 51-65.

Van Hooff, T., Blocken, B., & Van Harten, M. (2011). 3D CFD simulations of wind flow and wind-driven rain shelter in sports stadia: influence of stadium geometry. *Building and Environment*, *46*(1), 22-37.

Tominaga, Y., Mochida, A., Yoshie, R., Kataoka, H., Nozu, T., Yoshikawa, M., & Shirasawa, T. (2008). AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *Journal of wind engineering and industrial aerodynamics*, *96*(10), 1749-1761.

Ucuncu, M., Woolf D., & Zikri M. (2010). Thermal Comfort of Spectators in Stadia Built in Hot Climates. In *Proceedings of Conference: Adapting to Change: New Thinking on Comfort*. Cumberland Lodge, Windsor, UK, 9-11 April 2010.

Wolfram Incorporated, 2016. Wolfram Mathematica 11 Documentation.

Ward, G. J. (1994). The RADIANCE lighting simulation and rendering system. In *Proceedings of the 21st annual conference on Computer graphics and interactive techniques* (pp. 459-472). ACM.

White, A. J., Holmes, M. J., 2009. Advanced simulation applications using ROOM, *Eleventh International IBPSA Conference, Glasgow, Scotland*.