Prediction of Thermal Performance of Glass Roof Atriums using CFD Modelling

ASTHANA, Abhishek, SETHURAMALINGAM, Ramamoorthy, WESTON, Peter, GOEL, Mukesh and KANDASAMY, Sarvanakumar

Published version

ASTHANA, Abhishek, SETHURAMALINGAM, Ramamoorthy, WESTON, Peter, GOEL, Mukesh and KANDASAMY, Sarvanakumar (2019). Prediction of Thermal Performance of Glass Roof Atriums using CFD Modelling. In: AL-HABIBEH, Amin, ASTHANA, Abhishek and VUKOVIC, Vladimir, (eds.) The International Conference on Energy and Sustainable Futures (ICESF). Nottingham Trent University Pubilcations.

Copyright and re-use policy

Nottingham | UK | 9–11 September 2019

Prediction of Thermal Performance of Glass Roof Atriums using CFD Modelling

Abhishek Asthana, Ramamoorthy Sethuramalingam, Peter Weston, Mukesh Goel, Sarvanakumar Kandasamy

Hallam Energy, Sheffield Hallam University, United Kingdom

r.sethuramalingam@shu.ac.uk

Abstract

Glass covered or glazed atriums are becoming increasingly popular in public spaces for their aesthetics, quick installation and building energy reduction by taking advantage of natural daylight and heating by greenhouse effect. The estimation of the building load of atrium buildings is complicated due to the various thermal phenomena involved. The study aims to estimate the cooling loads or heat gain of glazed atriums, using computational fluid dynamics (CFD) modelling using an actual semi-open pedestrian walkway between two building blocks. Steady Reynolds Averaged Navier Stokes modelling approach with the RNG K-epsilon turbulence model, and the Discrete Ordinates (DO) Radiation models were used in the simulation. The temperature and air flow patterns predicted by the CFD simulations are discussed in this paper under various weather conditions. Results predict temperature gains inside the atrium, identify hot and cold spots and predict thermal comfort.

Keywords: Glazed Buildings, Building Thermal Performance, Computational fluid dynamics (CFD), Solar load

1. Introduction

High-glazed atrium-type spaces are becoming increasingly popular in public areas and buildings due to their architectural aesthetics as well as daylighting and solar heating advantages. The complex solar, airflow and thermal phenomena occurring in atrium spaces makes conventional load calculations difficult since they rely on assumptions of thoroughly mixed air and are thus inadequate to predict the thermal behaviour and energy performance in these spaces [1]. The well-stirred zone model is well applied to typical forced air system where relatively good air mixing is the design intent, but might cause unacceptable calculation errors for such system designs or operating modes as displacement ventilation, underfloor air distribution, chilled ceiling, natural ventilation, mix-mode ventilation, large spaces e.g., atria, auditoria, and so on, where nonuniformity of zone air temperature is designed intently to improve energy efficiency and indoor air quality [2]. It is of importance to consider the impact of nonuniform indoor air temperature on building load and energy use, which create a need for a different load calculation and system design method. Several researchers made efforts to find out a relatively accurate method for these particular spaces and systems. Beausoleil-Morrison [3] developed an adaptive controller to manage the interactions between the thermal and CFD modelling domains and implemented it within the ESP-r simulation program to support the conflation of CFD with dynamic whole building thermal simulation. Zhai et al [4] described several different approaches to integrating energy simulation and CFD and proposed a staged coupling strategy for different programs. Djunaedy et al [5] studied the implementation of external coupling between building energy simulation and CFD rather than a traditional internal coupling between the two different domains. Recently CFD strategies have been employed to investigate the natural ventilation air flow patterns and temperature gradient due to solar radiation [6]. Previous studies have confirmed that the temperature distribution and airflow velocity patterns predicted by the ANSYS Fluent turbulence models and solar radiation models were reasonably agreed with the experimental and analytical flow field data [6, 7]. The studies also discovered that as the solar intensity increases, the temperature inside the walkway increases too. In the same way, Ijaz et al. [8] evaluated the thermal comfort of the room using the ANSYS solver to support the natural ventilation for a school building in Singapore. Shafqat et al. [9] also investigated the CFD performance over the experimental work in natural convection with solar radiation exchange in an inclined solar plate heat exchanger using ANSYS DO irradiation model. The authors found agreement between numerical CFD results and experimental data in temperature and air flow velocity. A similar approach has been carried out in this present study but using the solar radiation as a heat source. The present study evaluates the thermal comfort in various weather conditions for the proposed walkway design using the validated CFD model procedures as referred by Halford [6] and Hunt, Shafqat Hussain [9].

2. CFD Methodology

The walkway temperature gradient and airflow pattern are governed by the conservation laws of mass, momentum, and energy equations in the CFD solver. These equations are standard and can be found in many textbooks and papers. Decomposing the Navier-Stokes equations (1) into RANS equations makes it easy to simulate realistic engineering problems in less computational time. A comprehensive description of these equations is provided in the papers by Hussain S et al., and Zhiyin Y [7, 10]. For example, the steady state equation is expressed below, [7] (1)

$$\frac{\partial(\rho u_i \phi)}{\partial x_i} = \frac{\partial}{x_i}(\Gamma_\phi \frac{\partial \phi}{\partial x_i}) + S_\phi \quad (1)$$

The mathematical model in the CFD includes numerical techniques to solve Reynolds Averaged Naiver Stokes equations (RANS) for incompressible three-dimensional turbulent flows. The commercial CFD code of ANSYS Fluent 18.0 solver for 3D double precision, pressure-based and the steady RANS equations have been used to solve the solution in combination with standard k–epsilon turbulence model. The governing equations of the solver include momentum, continuity and energy equations.

CAD model and Meshing

The 3-D model of the walkway has been constructed using the ANSYS design modeller. The CAD model, including brick works, granite slab works on the ground and the glass works of shops inside the walkway, has been constructed in the ANSYS software. Figure 3 illustrates the named surface materials in the 3D domain walls. Heights of the glass and bricks used in the walkway are depicted in figure 3: a); the glass wall is from 0 to 4.2 meters height from the ground, above 4.2 meters it is brick wall. In Figure 3: b), the brick wall is from 0 to 4.2 meters height from the ground and above 4.2 meters a combination of glass wall and brick wall is used. The westward (X) and northward (Z) lengths of the walkway are 101.4 meters and 84 meters respectively. The height of the canopy is 10 meters from the ground as shown in Figure 2. Prior to the CFD simulations, a mesh independence study was conducted over four mesh resolutions. Among four resolutions, results of 4.0 million mesh elements were proved to be sufficiently grid independent (Y+ = 1). A coarse structured mesh was used as shown in Fig.4.

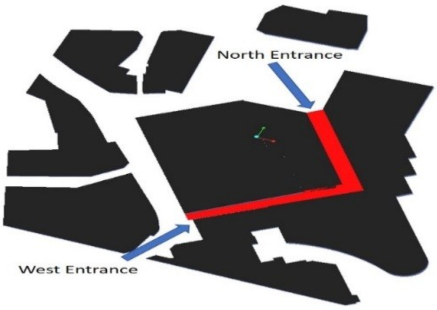


Figure 1: L-shaped walkway with nearby building

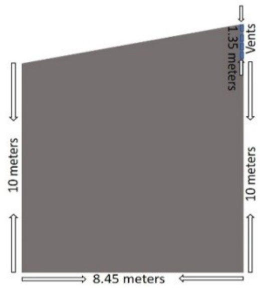


Figure 2: Cross section of the walkway

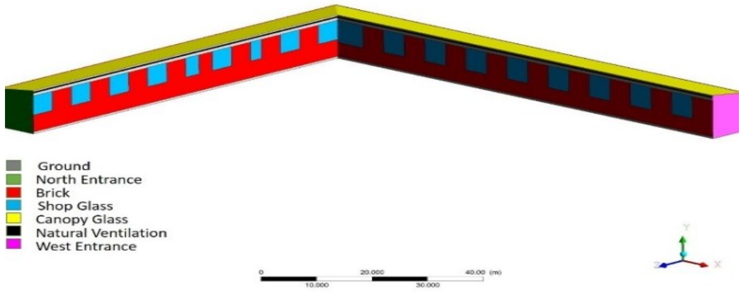


Figure 3: a) 3D view of the walkway with surface materials

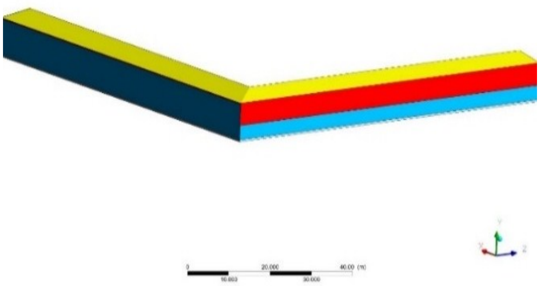


Figure 3: b) 3D view of the walkway

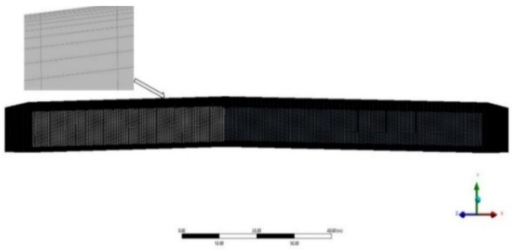


Figure 4: Structured mesh for the CFD model

Boundary Conditions:

Heat transfer through solar radiation is extremely important to consider in a naturally ventilated building. ANSYS Fluent Solver solves the radiation intensity transport equation which is derived from the solar radiation model. The outdoor wind speed and air through walkway entrances change the air flow pattern inside the walkway which affects the thermal sensation experienced by pedestrians inside the walkway. In this present study, the focus was to evaluate the thermal performance and thermal comfort of the building walkway during different weather conditions and at various wind speeds. Therefore, to determine the local wind speed and temperature data, government-supplied meteorological data is required. The Met Office's actual climatic data, such as temperature and sunshine factor, is collected from the nearest weather station (Ryhill, Met Office) and wind data from Emley Moor Met Office weather station for the proposed building. The conditions for various scenarios were selected according to the requirements of standards: ASHRAE 55 [11] and BS EN ISO 13792:2012 [12]. The conditions considered include average and peak day temperatures, normal direct solar irradiation and diffused horizontal irradiation, wind direction and speed at proposed location (Barnsley, UK). The ANSYS Fluent DO irradiation model was considered for this present study. The north entrance and ventilation in the walkway were considered as pressure outlets and the west entrance of the walkway was considered as a velocity inlet for windy conditions. The west entrance of the walkway was considered as a pressure outlet during the no wind condition. The level of comfort is often characterised using the ASHRAE thermal sensation scale. ASHRAE Index of PMV (predicted mean vote) is used to predict the sensation of pedestrians (+3 hot, +2 warm, +1 slightly warm, 0 neutral, -1 slightly cool, -2 cool, -3 cold) [13]. The material properties of bricks, glass and granite slabs are taken from the Heat Transfer Fundamentals book [14]. The transmissivity and absorptivity properties for the canopy's double-glazed glass are considered to be 0.70% and 0.35% respectively as per ASHRAE standards.

Table 1: Solar radiation and outside weather conditions as per fluent solar load calculator

Sun direction vector	**x- 0.7335 y- 0.3832 z- -0.56123**	
Month	June	October
Sunshine Factor	1	1
Direct normal solar irradiation (W/m^2)	1219.6	1257
Diffuse Solar irradiation – horizontal surface (W/m^2)	163.429	91.8

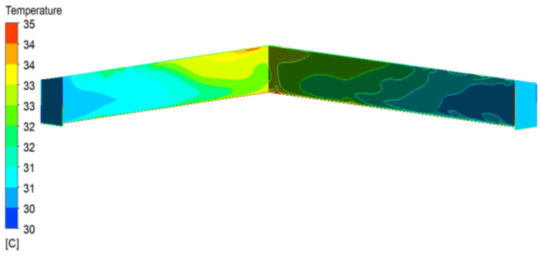
Table 1: Scenarios as per the standards suggested [11, 12]

Cases	Month	Ambient Temp(°C)	Day	Wind Speed(mph)	Weather Condition
1	Jun	31	D	0	Peak
2	Jun	31	D	0.2	Peak
3	Jun	31	D	5	Peak
4	Jun	20	D	1	Average
5	Oct	10	D	1	Average

3. Results and Discussion

A series of Computational Fluid Dynamics simulations were considered to investigate the temperature distributions inside the walkway. No wind condition and a series of windy conditions were studied in this paper. The comparative results reveal that in the critical scenarios, such as case 1, extra ventilation strategies are needed to reach neutral human comfort (-0.5 <PMV>+0.5). Increasing the solar intensity does increase the temperature inside the walkway. Figure 5 a) indicates that the stack pressure has been built up between the inside and outside of the building due to hot air generated inside the walkway. Similar phenomena were noticed in the study conducted by Hussain S et al. [7].

The overall results of the CFD simulations show that the natural convection process is not enough to reduce the heat built up inside the walkway to achieve the human comfort zone. Figure 5: a) illustrates the no wind scenario during peak summer weather conditions; there is a 4°C increase inside the walkway above the ambient temperature of 31°C. The buoyancy-driven steady state simulation was carried out in case 1 by utilizing the DO irradiation model in Fluent. Air is drawn from both opening entrances of the walkway and passes through the pedestrian walking space. Then the air is drawn out through the ventilation which is at the top of the walkway canopy. The driving force of the ventilation flow is assumed to be the temperature difference between the inside and outside of the walkway. A similar approach has been carried out to simulate the no wind scenario in a previous study [7]. The resultant PMV value of the walkway is thermally 'hot' in case 1. In the rest of the cases, the wind velocity was applied from the west side of the entrance.



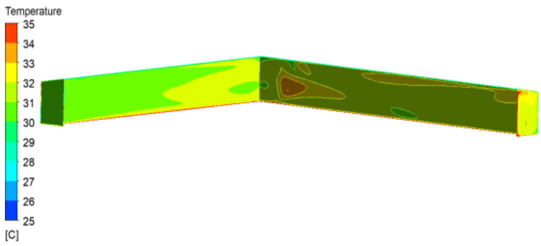
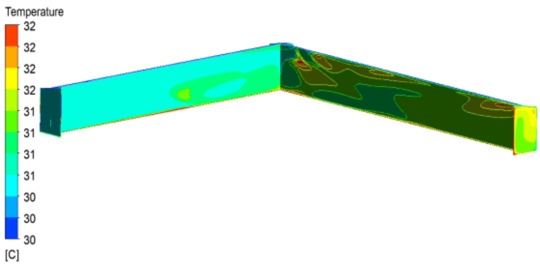
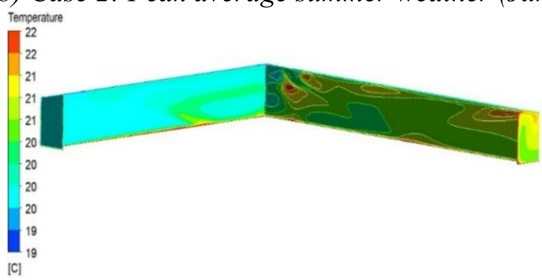
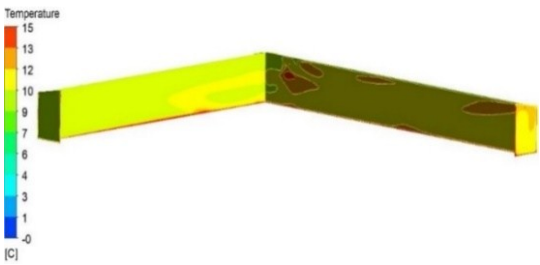


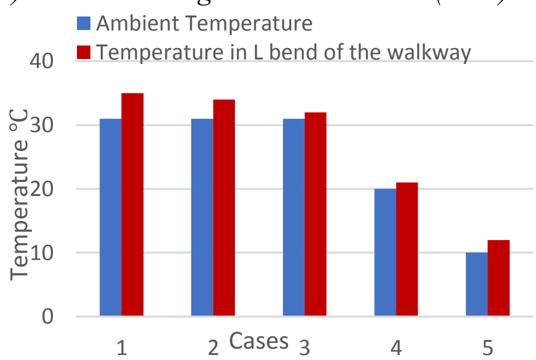
Figure 5:a) Case 1: Buoyancy driven flow (31,0mph) b) Case 2: Peak average summer weather (June)





c) Case 3: Peak average weather (June, 5mph) *d) Case 4: Average summer weather (June)*





e) Case 5: Autumn average weather (October) *Figure 6: Summary of temperature rises in all 5 cases*

In Figures 5: b) and c) the maximum temperature rise seen near the L-bend of the walkway is 3°C and 1°C when there is wind velocity of 0.2 mph and 5 mph respectively. As in case 1, the resultant PMV value is thermally 'hot' in case 2. Alternatively, the resultant PMV value of the walkway in case 3 is thermally 'natural' which indicates the key role of wind velocity in thermal comfort. Similar behaviour was also detected in the study by Ijaz Fazil S et al. [8]. Figure 5: d) illustrates the average temperature in June at a wind speed of 1 mph. It can be noted that the temperature inside the walkway increases by 1°C above the ambient temperature (20°C) due to solar intensity. The resultant PMV value of the walkway is thermally 'slightly cold'. Figure 5: e) represents the temperatures at 10°C during the spring and autumn months with a wind speed of 1 mph. It is seen that the temperature inside the walkway increased by about 2°C above the ambient temperature of 10°C. The resultant PMV value of the walkway is thermally 'cold'. Like the study by Hussain S eta al., [7] the solar intensity and wind speed played key roles in the temperature rise and decrease in the present study. A summary of temperature rises in all 5 cases are presented in Figure 6. Figure 6 illustrates all temperature data collected from the probe located in the centre of the L–bend at 1.75 meters height from the ground. In cases 1 to 3, as the wind velocity increases, the temperature inside the walkway decreases. Similar behaviour was shown in the study conducted by Ijaz Fazil S et al. [8].

4. Conclusion

The study results recognized that the ambient temperature and wind speed have a greater influence in the temperature distribution inside the walkway. In hot weather, wind will play a key role in removing heat from the walkway. Similarly, inclined solar radiation plays a key role in determining the temperature distributions inside the walkway. CFD techniques developed in this study can be employed in the initial design stage of naturally ventilated buildings to promote human thermal comfort and its energy efficiency. The future work of this research will include the experimental validation against the numerical work. Overall, the methodology developed in this study can be used as a procedure for construction firms to evaluate energy-efficient building designs.

Acknowledgements

This work is supported by Barnsley Council and Hendry Boot Construction Ltd.

References

1. Griffith B., Chen Q (2004) Framework for coupling room air models to Heat Balance Model Load and Energy Calculations. HVAC&R RESEARCH, Vol 10, No 2.

2. Yiqun Pan, Yuming Li, Zhizhong Huang (2009), Study on Energy Modelling Methods of Atrium Buildings, 11th International IBPSA Conference, Glasgow, July 27-30.
3. Beausoleil-Morrison I (2001) The adaptive coupling of computational fluid dynamics with whole-building thermal simulation. Building Simulation. Brazil.
4. Zhai Z., Chen Q., Klems J. H., Haves P (2001) Strategies for coupling energy simulation and computational fluid dynamics programs. Building Simulation. Brazil.
5. Djunaedy E., Hensen J.L.M., Loomans M.G.L.C (2005) External coupling between CFD and Energy Simulation: implementation and validation. ASHRAE Transactions, Vol. 111, Part 1.
6. Holford, Hunt et al (2003) Fundamental atrium design for natural ventilation. Building and Environment 38: 409-426
7. Hussain S, Patrick H (2013) Numerical study of buoyancy- driven natural ventilation in a simple three storey atrium building. International Journal of Sustainable Built Environment 1:141-157
8. Ijaz Fazil S A K and Sivamoorthy K et al (2017) Performance evaluation of airflow and thermal comfort in the room with wind catcher using different CFD techniques under natural atmosphere boundary layer. Applied Energy Symposium & Forum: Low Carbon Cities & Urban Energy Joint Conference 143: 199-203
9. Shafqat H (2017) Natural ventilation of natural convection and radiation exchange in an asymmetrically heated inclined channel. Imam Journal of Applied Science Volume -1
10. Zhiyin Y (2014) Assessment of unsteady RANS approach against steady RANS approach for predicting twin impinging jets in a cross flow. Cogent Engineering 1: 936995
11. ASHRAE 55 (2017) Thermal environmental conditions for human occupancy. American National Standard Institute ISSN 1041: 2336
12. BS EN ISO 13792 (2012) Thermal performance of the building – calculation of internal temperatures of a room in summer without mechanical cooling – simplified methods. BSI Standard Publication ICS 91.120.10
13. Juyoun K, Ken P (2008) Evaluation of the PMV thermal comfort index in outdoor weather conditions. Environmental Ergonomics Research Centre, Loughborough University.
14. Yildiz B, Necati O (2012) A Textbook for Heat Transfer Fundamentals. Begell, Danbury.