Albatayneh, Aiman; Alterman, Dariusz; Page, Adrian. "Adaptation the use of CFD modelling for building thermal simulation". Published in ICSIM2018: Proceedings of the 2018 International Conference on Software Engineering and Information Management (Casablanca, Morocco 04-06 January, 2018) p. 68-72 (2018).

Available from: http://dx.doi.org/10.1145/3178461.3178466

Adaptation the Use of CFD Modelling for Building Thermal Simulation

Aiman Albatayneh
School of Natural Resources Engineering and Management, German Jordanian University, Amman, Jordan P.O. Box 35247.
+962-06-4294444 Ext; 4218
Aiman.Albatayneh@gju.edu.jo.

Dariusz Alterman
Priority Research Centre for Frontier Energy Technologies and Utilisation, The University of Newcastle, NSW, Australia.
+61-02-0339330
dariusz.alterman@newcastle.edu.au.

Adrian Page
Priority Research Centre for Frontier Energy Technologies and Utilisation, The University of Newcastle, NSW, Australia.
+61-02-49216587
adrian.page@newcastle.edu.au.

ABSTRACT

This paper demonstrate the possibility of using CFD simulation alone to determine the internal air temperature of buildings for long periods (one year), without the assistance of any additional software, with fast computing times and an acceptable degree of accuracy for the simulation results.

An experiment on CFD simulations were carried out to examine the accuracy of CFD simulation to predict building internal air temperature for a complete house in Perth, Australia. Real data recorded inside a house were compared with CFD modeling results to find the precision of the CFD simulation after CFD adaptation process applied in this research.

This study is an attempt to use CFD alone to calculate the buildings internal air temperatures for extended periods (months, years). Using CFD simulations with extended periods have some problems, for example; lengthy computing times, discrepancies in peak temperature time and the internal air temperatures inside the model keep rising with time. Performing CFD analysis after applying the measures to adapt the use of CFD modeling resulted in faster computing times, with 1% of the computing time compared to that for a 1 minute time step, and with 90% of the results lying within 3°C of the real (observed) data. The overall results from CFD simulations with an average accuracy of 92% compared with the real data recorded inside the house.

CCS Concepts

Computing methodologies massively parallel and high-performance simulations.

Keywords

Internal Air Temperature; CFD; Building Simulation; Thermal Comfort.

1. INTRODUCTION

Computational Fluid Dynamics (CFD) is a modeling technique that simulates fluid flow and heat transfer mathematically. CFD is a powerful tool that creates a virtual airflow and building thermal model to evaluate and optimize a design before construction begins. Changes to an existing building can also be assessed using CFD prior to any renovations. This method has the advantages of decreasing design risks and avoiding costly inaccuracies while allowing innovations and improvements [1].

CFD has been playing an increasingly important role in building design by contributing to achieving a comfortable, healthy, and energy-efficient building design. CFD can be used in a wide range of applications in building design, from building site layout design to individual room planning. It can also be used for active heating, ventilating and air-conditioning (HVAC) system design through to passive ventilation studies, and from consistent indoor air quality valuation to serious fire smoke and toxin control [2]. CFD has been playing important role in building design, following its continuing expansion for decades. Some of these energy simulation programs BES coupled with computational fluid dynamics where CFD simulations use a small time-step and BES handles a long-term simulation [3,4,5].CFD were used to investigate the accuracy of gas dispersion in the vicinity of buildings and complex terrain by comparing predictions with wind-tunnel measurements which show some concerns about the accuracy of the CFD modeling [6].

The results provided by CFD can be used to analyze whole buildings to find the internal air temperature at any point inside the building but using CFD for long term simulations results in excessive computing time.

Autodesk Simulation CFD can be used throughout the design process by systematizing the procedures by applying settings directly in the CAD model and then connecting (export) CAD model designs in the Autodesk Simulation CFD to designs, studies and analysis. Throughout the design, development and marketing processes of any project CFD can be used [7]:

- In the concept stage, CFD can be used to try new ideas and designs, optimize various scenarios, understand the implications of the concept before starting the design process and determine how it will perform in the real world.
- In the development stage, CFD can be used to analyze the effects of design changes to reach a final design that brings the projected performance.
- For the marketing stage, CFD can be used to visualize the products/ideas by creating 2D and 3D images and videos that show the design assets and advantages.

There are two main methods to analysis the gradients, i.e. CFD and Finite Element Analysis (FEA). The FEA method is usually used in the stress concentration analysis whilst the CFD method simulates fluids performance including heat transfer, combustion and thermodynamic effects. The main reason of selecting CFD was the nature of the problem which involves the analysis of solar radiation, humidity, air temperature and movements. Although, FEM may be used for the analysis of the temperature gradient for solids (i.e. walls, slab and roof) but it would be difficult to accurately simulate the convection effects of the air. CFD is well recognized in range of applications in building design, from building site layout design to individual room planning. It has been used in various applications where fluid analysis was required such as active heating, ventilation and air-conditioning system design [2].

CFD can be used in a wide range of building design, such as:

- Site Planning

The first phase of building design is site planning. CFD can assist in optimizing building sites by calculating humidity, temperature, air velocity, turbulence concentrations and pollutant concentrations surrounding the buildings. It can also improve energy efficiency by optimizing passive HVAC by using natural ventilation for summer. The main obstacle of CFD is the lengthy computing time due to the great number of mesh grids and nodes essential to shield the design structure and the computing time becomes greater when dynamic wind conditions are applied [2].

Natural Ventilation Analysis

Optimizing natural ventilation is one of the most essential methods to decrease energy consumption in buildings (in summer, by allowing cool air to enter the building and, in winter, by preventing cold air from inflowing). CFD can simulate buildings internal airflows to reach optimum natural ventilation.

- HVAC System Design

CFD was used to assess the thermal comfort and air quality generated from various HVAC models to reach efficient and operational system designs. It is more comprehensive than the conventional analysis methods which depend on the use of various companies' charts and equations [2].

- Regulate Pollution and Dust

CFD was able to simulate the movements of pollutants with a high degree of accuracy and low costs. Also it can be effortlessly applied to examine the effect of a specific flow factor (air temperature or wind speed) or the scattering of a specific pollutant in extremely hot conditions or contaminated situations [2].

The main issues with CFD analysis for prolonged simulation are: its lengthy computing time [8,9,10,11]; internal air temperature increases with longer simulation periods (warming up); smaller fluctuation range and discrepancies in peak temperature time. This research will find new ways to solve these issues to accurately simulate the thermal performance of buildings for long periods (weeks and months).

2. METHODOLOGY

The Western Australian house used in the study is located near Perth, where the weather is warm and sunny most of the year. This climate zone is unique in Australia because it has more sunny days per year than any other Australian capital city. In the summer months (December - February) the weather is warm to hot, and the hottest months are usually January and February. Autumn (March - May) has warm sunny days and cooler nights. Winter (June - August) is the rainy season, with cool sunny days. In spring (September - November), the days are warm and sunny [12].

The house, as shown in Figure 1, is located in Perth, Western Australia, Australia (Latitude = -32.131166, Longitude =116.000648). The closest weather station recording air temperature every minute is Jandakot Aero (Station numbers: 009172).

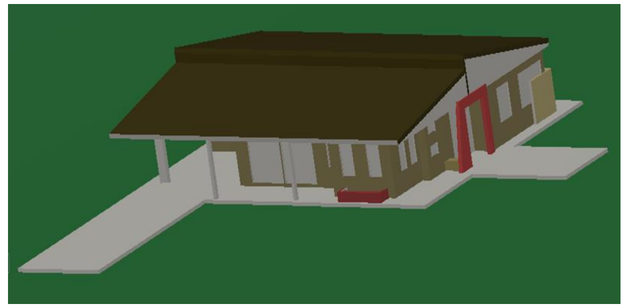


Figure 1. Western Australian house.

The building was not occupied and no heating or cooling systems were operated; the windows were closed during the studied period. The house has three bedrooms and one activity area, an office, kitchen and family room, as shown in Figure 2.

The total area of the house is 168.4 m^2 (160.1m^2 air conditioned and 8.3m^2 unconditioned), the house was built using insulated cavity brickwork for the external walls with bulk insulation R 1.5, and the internal walls were built using single skin brickwork. 18o Hip and Gale were used for the roof with a volume of 198m^2, and plasterboard bulk insulation R2.5 was used for the ceiling. There is a concrete slab on the ground for the floor with no insulation. Sensors were installed in each room inside the house to record the temperature every 5 minutes over one year.

Computational Fluid Dynamics is an established simulation technique that mathematically simulates fluid flow and heat transfer. CFD analysis can be used to analyze whole buildings to find the internal air temperature at any point inside the building. CFD is not specifically designed to calculate a house internal air temperature over long periods. It can be used for this purpose but lengthy computing times are required [12]. Usually, CFD is used to evaluate a building's short term thermal performance and coupled with other programs to predict prolonged performance, such as Building Energy Simulation (ES), to avoid this excessive computing time.

The variations of internal air temperature of the house for different rooms can be compared to the simulated CFD results, in order to determine the accuracy of the CFD simulations. The CFD modeling then can be used to find the average volume temperature inside each room.

The house was analyzed using Autodesk Simulation CFD 2014. The CFD models were modeled based on the actual physical characteristics and properties of the house to obtain results as close as possible to those recorded from the house. The simulations were based on the measured outside air temperature and the wind speed and direction recorded from the house. The simulation results were then compared with the actual air temperature recorded inside the house.

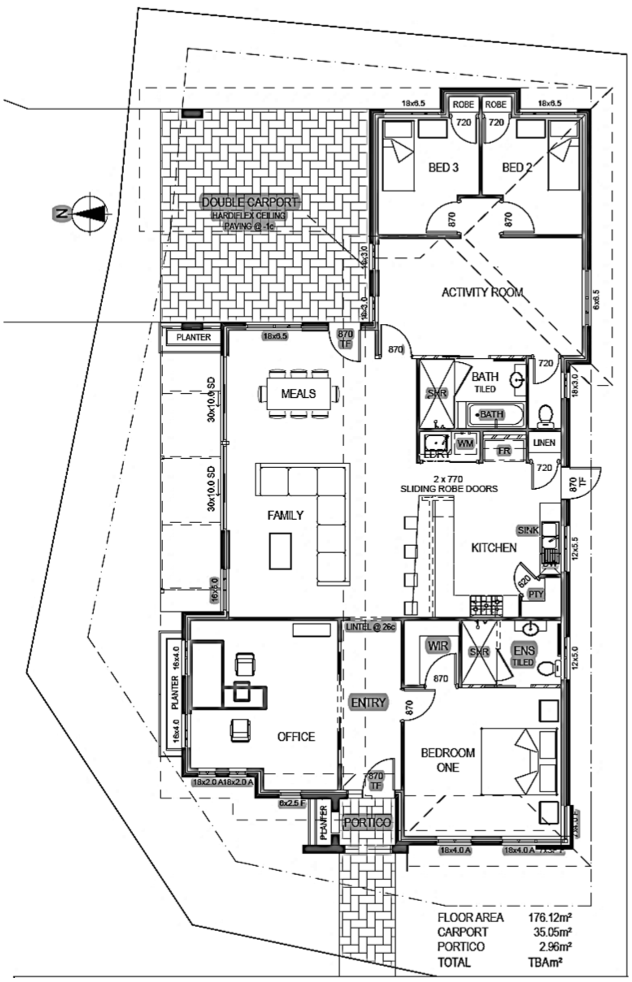


Figure 2. Floor plan for WA house.

Autodesk Simulation CFD software turns a 3D CAD workstation into a fully interactive flow workbench. 3D gatherings become zero-cost prototypes showing critical engineering information and data not obtainable from physical experiments. Immediate results are obtainable if any changes take place to the design (Autodesk 2014).

General steps for an Autodesk CFD building simulation are (Autodesk 2014):

- Create CAD drawing for the real house using Inventor Fusion 2013 R1 then launch/export it into the Autodesk Simulation CFD.
- Use geometry to perform these key functions: set the analysis length units, state the coordinate system for the 2D house and access the geometry tools.
- Allocate materials to all parts in the house and assign boundary and initial conditions.
- Generate the mesh where the geometry is divided into small pieces called elements and the corner of each element is a node. These elements and nodes form the mesh.
- Start to solve the analysis using different time steps, where smaller time step sizes take a longer computing time.
- Results used visualization tools to help monitor, analyze, and present the results.

The geometrical characteristics for the house and their material properties were modeled using Inventor Fusion 2013 R1 then launched into the Autodesk Simulation CFD. A large external environment of a 100m x 100m x 100m external volume, in the shape of a cube to surround the building, was constructed in the CFD. Then the material properties the house was assigned with the same thermal properties as the real house.

A transient temperature boundary condition was applied to the surface of the external volume surrounding the house and initial conditions applied, based on real data collected from the house.

A transient solution mode, heat transfer, flow and radiation were enabled and calculated in the CFD simulation software by entering the exact location and date of the real house. The solar heating function was also enabled with the latitudinal and longitudinal position of the house reflecting their locations. An appropriate date, time and orientation were also entered to reflect the real conditions.

3. RESULTS AND DISCUSSION

The performance of a three bedroom house in suburban Perth, Western Australia is assessed. CFD simulations were used to find the building's internal air temperatures and were compared to the results with the real data obtained from the site in order to assess the accuracy of the CFD simulations.

By having external and internal data over one year, the observed internal air temperature for the house can be compared with the simulated CFD results. However, when compared with the observed data, CFD simulations for long periods face some potential issues which need to be addressed:

- Air temperature inside the building keeps warming with time
- Excessive computing time
- Smaller fluctuation range
- Discrepancies in peak temperature time.

Adding the new boundary condition (new volume / external air layer enclosing the simulated module) the prolonged rise in the internal air temperature for the simulated modules was prevented. The CFD simulations with two external air boundary conditions provided more accurate results compared to the one external air boundary condition. It also provided a better representation of the real air temperatures inside the module, where more than 89% of results of the CFD simulated modules with two external air boundary conditions falling within 0 - 3ºC temperature difference compared with the real air temperatures measured inside the module [13].

Larger time steps were used to speed up the simulation time and give higher temperature fluctuation ranges but they increased the discrepancies in peak temperatures (the peak temperature inside the building will shift/lag compared with the real data). There is a direct relationship between the time step size and the temperature fluctuation range, where the temperature fluctuation range increased with a larger time step size [14]. The speediest CFD simulations with best accurate results came from the 80/100 time step when 90% of the results within 3ºC of the real data, with 1%

of the computing time required compared to the 1 minute time step [15].

CFD simulations were carried out where the temperature variations outside the house changed with the time of the day and the season, as shown in Figures 3.

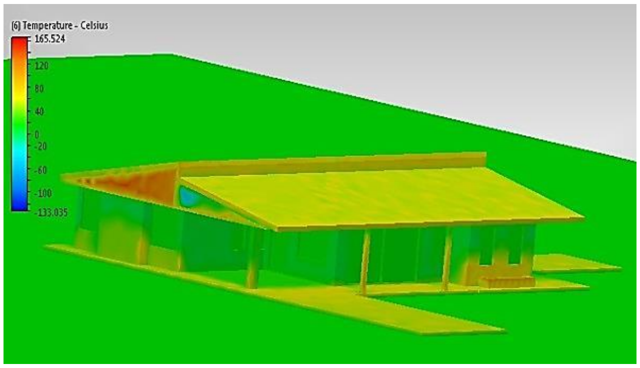


Figure 3a. Snapshots from the CFD simulations showing temperature variations on the external surfaces during sunny morning spring day (15/10/2009).

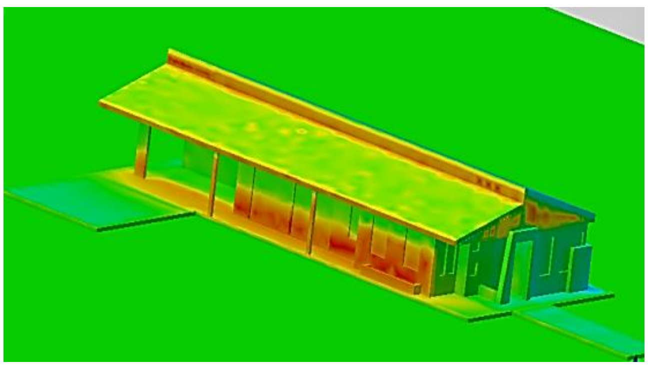
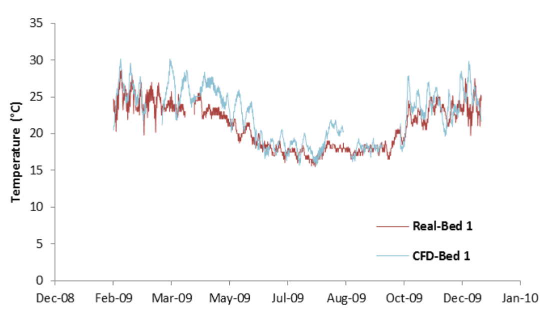
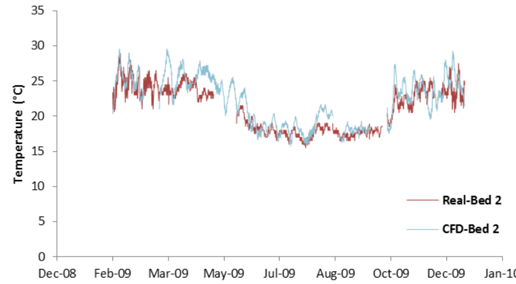


Figure 3b. Snapshots from the CFD simulations showing temperature variations on the external surfaces during the evening for sunny winter day (18/06/2009).

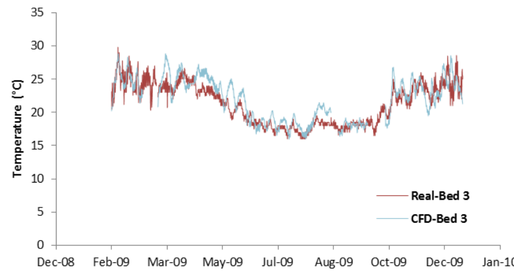
To determine the accuracy of the CFD simulations, the variations of the internal air temperatures for each room between the simulated CFD results and the real house data were compared, as shown in Figures 4.



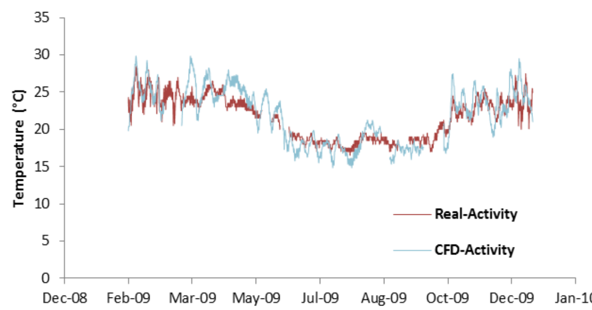
Figures 4a. Variations of the internal air temperatures between the CFD simulation results and the real data for bedroom 1.



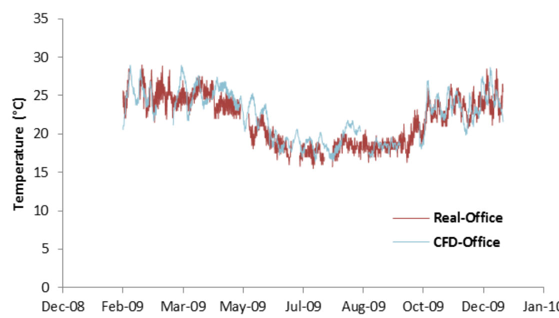
Figures 4b. Variations of the internal air temperatures between the CFD simulation results and the real data bedroom 2.



Figures 4c. Variations of the internal air temperatures between the CFD simulation results and the real data for bedroom 3.



Figures 4d. Variations of the internal air temperatures between the CFD simulation results and the real data for the activity room.



Figures 4e. Variations of the internal air temperatures between the CFD simulation results and the real data for the office.

Comparison between the CFD simulations and the real data for each room inside the house showed that the average accuracy, at any given time during one year's simulation, was around 92.26% for all rooms, as shown the Figure 5.

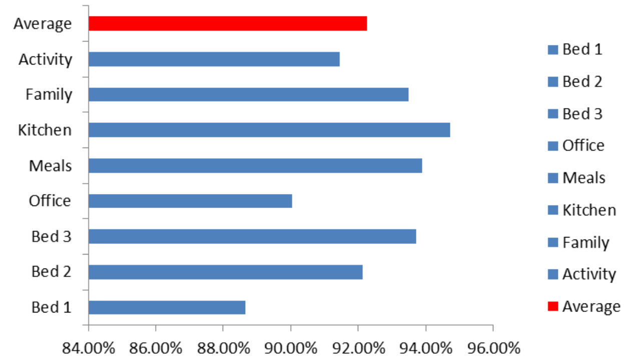


Figure 5. The average accuracy (%) between the real data and the CFD simulated data during the simulation period.

Comparison between the CFD simulations with the real data for a typical house at any given time during one year's simulation indicated that the average accuracy was in the order of 92%.

4. CONCLUSIONS

To illustrate the applicability of the adjustments made for the CFD analysis for the complete structure, the analysis of actual Western Australia house to enable the use of CFD analysis for prolonged simulations which have been developed and described in this paper. CFD simulations over long periods face issues, such as: the simulated internal air temperature warming with time; long computing times; smaller fluctuation range and discrepancies in peak temperature time.

Adding a new boundary condition/layer surrounds the house with the ambient external air temperature without affecting the solar radiation received by the house. This new boundary layer prevented the long term rise in the internal air temperature, which resulted in more than 89% of the results falling within 0 - 3ºC temperature difference compared with the real air temperatures measured inside the house. Larger time steps were used (80/100 minutes) in the CFD analysis to speed up the simulation time by 99% compared to 1 minute time step computing time. Performing CFD analysis after applying the measures resulted in faster computing times, with 1% of the computing time compared to that for a 1 minute time step, and with 90% of the results lying within 3ºC of the real (observed) data. After solving main CFD simulating issues, simulations were carried out for a house for a one year period, with an average accuracy of 92% when compared with the real data at any given time during the year. These results are encouraging and may enable the use of CFD simulation as an innovative building modeling tool to precisely calculate the internal air temperature for any building envelope.

5. ACKNOWLEDGMENTS

The authors acknowledge the financial support of the Deanship of Scientific Research at the German Jordanian University for the presented work in this paper.

6. REFERENCES

[1] BMT (2017). Built Environment CFD. [online] Bmtwbm.com.au. Available at: http://www.bmtwbm.com.au/markets/water-and-environment/environmental-cfd/built-environment-cfd/ [Accessed 24 Oct. 2017].

[2] Zhai, Z., 2006. Application of computational fluid dynamics in building design: aspects and trends. Indoor and built environment, 15(4), pp.305-313.

[3] Yi, Y.K. and Feng, N., 2013, September. Dynamic integration between building energy simulation (BES) and computational fluid dynamics (CFD) simulation for building exterior surface. In Building Simulation (Vol. 6, No. 3, pp. 297-308). Tsinghua University Press.

[4] Zhai, Z. and Chen, Q.Y., 2003. Solution characters of iterative coupling between energy simulation and CFD programs. Energy and buildings, 35(5), pp.493-505.

[5] Zhai, Z., Chen, Q., Haves, P. and Klems, J.H., 2002. On approaches to couple energy simulation and computational fluid dynamics programs. Building and Environment, 37(8), pp.857-864.

[6] Robins, A.G., Hall, R., Cowan, I.R., Bartzis, J.G. and Albergel, A., 2000. Evaluating modelling uncertainty in CFD predictions of building affected dispersion. International Journal of Environment and Pollution, 14(1-6), pp.52-64.

[7] Help.autodesk.com. (2017). Help. [online] Available at: http://help.autodesk.com/view/SCDSE/2014/ENU/?guid=GUID-C1E9EBEA-B7DF-478A-A795-24F337C93DF3 [Accessed 24 Oct. 2017].

[8] Hong, T., Chou, S.K. and Bong, T.Y., 2000. Building simulation: an overview of developments and information sources. Building and environment, 35(4), pp.347-361.

[9] Hulme, I. and Kantzas, A., 2005. Validation of bubble properties of a bubbling fluidized bed reactor using CFD with imaging experiments. Polymer-Plastics Technology and Engineering, 44(1), pp.73-95.

[10] Tan, G. and Glicksman, L.R., 2005. Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. Energy and Buildings, 37(10), pp.1049-1057.

[11] Wang, L. and Wong, N.H., 2009. Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment. Building and Environment, 44(1), pp.95-112.

[12] Perth Weather, Climate & Temperature - Tourism Australia, Australia.com. (2017). [online] Available at: http://www.australia.com/en/facts/weather/perth-weather.html [Accessed 24 Oct. 2017].

[13] Albatayneh, A., Alterman, D., Page, A. and Moghtaderi, B., 2015. The significance of time step size in simulating the thermal performance of buildings. Advances in Research, 5(6), pp.1-12.

[14] Albatayneh, A., Alterman, D., Page, A. and Moghtaderi, B., 2017. Discrepancies in Peak Temperature Times using Prolonged CFD Simulations of Housing Thermal Performance. Energy Procedia, 115, pp.253-264.

[15] Albatayneh, A., Alterman, D., Page, A.W. and Moghtaderi, B., 2016. WARMING ISSUES ASSOCIATED WITH THE LONG TERM SIMULATION OF HOUSING USING CFD ANALYSIS. Journal of Green Building, 11(2), pp.57-74.