Alternative Method to the Replication of Wind Effects into the Buildings Thermal Simulation

Aiman Albatayneh [1],* [ID]**, Dariusz Alterman [2], Adrian Page [2] and Behdad Moghtaderi [2]**

[1] Department of Energy Engineering, School of Natural Resources, German Jordanian University, P.O. Box 35247, Amman 11180, Jordan

[2] Priority Research Centre, The University of Newcastle, Callaghan, NSW 2308, Australia; Dariusz.Alterman@uon.edu.au (D.A.); Adrian.Page@uon.edu.au (A.P.); Behdad.Moghtaderi@uon.edu.au (B.M.)

* Correspondence: Aiman.Albatayneh@gju.edu.jo

check for updates

Abstract: To design energy-efficient buildings, energy assessment programs need to be developed for determining the inside air temperature, so that thermal comfort of the occupant can be sustained. The internal temperatures could be calculated through computational fluid dynamics (CFD) analysis; however, miniscule time steps (seconds and milliseconds) are used by a long-term simulation (i.e., weeks, months) that require excessive time for computing wind effects results even for high-performance personal computers. This paper examines a new method, wherein the wind effect surrounding the buildings is integrated with the external air temperature to facilitate wind simulation in building analysis over long periods. This was done with the help of an equivalent temperature (known as $T_{natural}$), where the convection heat loss is produced in an equal capacity by this air temperature and by the built-in wind effects. Subsequently, this new external air temperature $T_{natural}$ can be used to calculate the internal air temperature. Upon inclusion of wind effects, above 90% of the results were found to be within 0–3 °C of the perceived temperatures compared to the real data (99% for insulated cavity brick (InsCB), 91% for cavity brick (CB), 93% for insulated reverse brick veneer (InsRBV) and 94% for insulated brick veneer (InsBV) modules). However, a decline of 83–88% was observed in the results after ignoring the wind effects. Hence, the presence of wind effects holds greater importance in correct simulation of the thermal performance of the modules. Moreover, the simulation time will expectedly reduce to below 1% of the original simulation time.

Keywords: building simulation; Computational Fluid Dynamics (CFD); wind effect; building thermal performance; low-energy buildings

1. Introduction

The building sector is responsible for a high level of greenhouse gas (GHG) emissions, because of its tremendous energy consumption. Therefore, reducing energy consumption in the building is an important strategy to mitigate climate change and reduce GHG emissions. Accurate estimation of internal air temperature is necessary to limit the energy consumption and simultaneously ensure sustained thermal comfort; this needs appropriate building thermal simulation models due to the ability to minimize future energy consumption in the residential sector. The building energy assessment programs need to be implemented for designing energy-efficient buildings. Determining the internal air temperature of the building is the ultimate objective, since the energy required for the provision of thermal comfort to occupants can be computed through this key factor. Hence, the experts will require an accurate building thermal simulation modeling software, so that the greenhouse gas

(GHG) emissions as well as the operational energy requirements of the building can be reduced to a great extent.

Buildings with increased energy efficiency are critical for energy usage in the future. Innovative designs with enhanced creativity should be developed and implemented in the construction industry to optimize thermal performance. In order for this to be achieved, it is necessary to realistically predict building performance according to a variety of conditions, which can be performed by utilizing effective instruments including assessment software programs. Despite the existence of such programs, their effectiveness has been relatively limited thus far. It is important to consider all physical, environmental, and social aspects prior to the construction of a building. This will facilitate improved prediction of the building's energy usage and lead to a reduction in total operational costs across the lifespan of the building [1].

Hence, the precision of a given thermal simulation will directly impact the ability to predict the operational energy costs for a building's lifetime. Computational fluid dynamics (CFD) is a powerful instrument whose role in the design of energy-efficient buildings is gradually increasing. The role of CFD in the process of designing buildings has assumed increased importance in recent years as a result of its ongoing evolution for over 25 years. The data delivered by CFD can be utilized for analyzing the thermal performance of the building. It is possible to use CFD for a wide variety of different applications in the building design process, including the overall layout design of the construction site and the planning of individual rooms. Additionally, it can be employed for the design of active heating, ventilation and air-conditioning systems. After a decades-long process of ongoing development, CFD can now be utilized for analyzing the inside air temperature in any location inside the building space. One of the benefits of Autodesk Simulation software is that it decreases the design risks and prevents expensive miscalculations while promoting innovation and easy enhancements. CFD is capable of converting any Computer-Aided Design (CAD) drawing (a method frequently employed by the designers of buildings) into zero-cost prototypes with complete interactivity that show immediate critical engineering data and information that cannot be accessed from actual experiments.

The building materials and layout, wind speed and direction, location, sun availability, thermal mass, and orientation are the fundamental elements influencing the thermal performance of the building [2]. The heat transfer between the building and its surroundings is the primary element required for the consumption of energy inside the building, which takes place through conduction, convection, and radiation through walls, roof, windows, and floor. The wind speed is the major factor affecting convection heat transfer that must be integrated in any simulation tool.

Though this could lead to extended simulation intervals, by using different assumptions, quantitative and qualitative analyses of the wind effect could be performed through various simulation tools [3,4]. The building thermal analysis and the energy required to maintain occupants' thermal comfort would be affected by the impact of different simulation hypotheses [5]. Thermal comfort in the residential buildings could be achieved if the design of wind ventilation is most appropriate and optimal [6]. This relies on the wind direction, i.e., the wind comfort around buildings is improved through the lift-up design and the effects are usually determined by the approaching wind directions [7].

There are different software packages used in building thermal simulation [7], for example, the expertise of TRNSYS (Transient System Simulation Tool, developed at the University of Wisconsin, USA) can be utilized in complete solar energy system modeling, while application of CFD can be observed in building design from building site layout design to individual room planning. Moreover, it can also be used for active Heating and passive Ventilation study to ventilating and Air-Conditioning (HVAC) system design, and from serious fire, smoke, and toxin control to consistent indoor air quality valuation [8].

Autodesk Computational Fluid Dynamics (CFD by Autodesk) is one of the leading software packages. It has remained successful, witnessed continuous improvements, and the users are likely to perform buildings' thermal analyses through this software to determine internal air temperature and buildings' energy consumption. To perform the simulation, sky temperature is needed by the CFD

analysis. Hence, the accuracy of the CFD analysis will be subject to the accuracy of the sky temperature. Moreover, the privilege of importing files directly from AutoCAD to CFD is one of the key benefits of CFD over TRNSYS, which is considerable for building design, as building designers extensively use AutoCAD [9].

The green building design and construction can be enabled through CFD, which has the potential to generate a building thermal model and a virtual airflow to evaluate the design prior to the start of construction. The overall goal is to achieve a healthy, comfortable, and energy-efficient building design. Before any renovation works, CFD can be used to assess the likely changes to an existing building. Through this method, the users can realize benefits of moderated design risks besides preventing overpriced errors, while enabling improvements and revolutions [10]. There are certain shortcomings while using CFD for regular thermal simulation of buildings (months and up to one year), for example, inconsistencies in peak temperature times using prolonged CFD simulations [11], warming issues connected with the ongoing simulation [12], and time step size and simulation [13] of wind impacts on the thermal performance of buildings [14].

By and large, the simulation tools are devoid of certain information, meaning they fail to capture the real thermal performance of the buildings [15]. Variances between projected theoretical energy loads and the actual energy consumption are produced by most building thermal assessment programs [16]. For long time periods, the internal air temperature of buildings can be simulated by the CFD analysis; however, this requires lengthy computing times [17]. Therefore, building simulations are carried out through Energy Simulation Software (BES) coupled with CFD analysis, where the long period simulations are executed through BES, while short period simulations are executed through the CFD [18,19].

Mainly, three methodologies are used by the CFD analysis to forecast turbulent flows. These three are large-eddy simulation (LES), direct numerical simulation (DNS), and Reynolds-averaged Navier-Stokes (RANS) equation simulation with turbulence models. Minor time steps are required by the DNS model, leading to a long and sustained simulation [20]. The dynamic wind conditions may result in significant computing time. This will still need considerable computing time even with powerful computers [21].

In certain thermal modeling programs, building energy simulation is combined with CFD analysis; nevertheless, while CFD incorporates a small time step, BES focuses on the long-term simulation. The calculation of the internal air temperatures of a building for long-term simulations by utilizing only CFD would necessitate a long period of computation. In the past, examination of the majority of issues pertaining to buildings such as ventilation analysis, wind loading, and wind environment, among others, was performed with wind tunnel tests; however, it is now possible to effectively conduct such tests using CFD. CFD has the ability to find solutions to the aforementioned problems using an approach that is more powerful and economical than the previous one. Today, computational fluid dynamics is employed as a state-of-the-art technique for modeling airflow and is effective at predicting airflow, heat transfer as well as the transportation of pollutants into and around structures.

2. Issues with Wind Simulations

Building thermal simulation might encounter various issues, for example, extreme computing time, divergence, and time step size.

2.1. Time Step Size

Regarding CFD analysis, the type of analysis and the time scale of the analysis are the required parameters for time step size. A fraction of the mean flow velocity for non-motion flow analyses is none other than the time step size, and it should be a tenth of the time needed to cross the length of the object under consideration. Adequately resolving the flow will require a much smaller time step size (1/20th) in many scenarios [22]. For instance, the solver can edit the time step size in Autodesk

CFD Simulation (2014) to terminate the transient analyses either upon attaining an exact time or after a certain number of time steps (whichever comes first).

Concerning the time step size, a handy guideline is nearly 1/20th of the time required for a fluid to pass through the length of the object. For instance, for a 6 m long building (*L*), there should be an average wind speed (*V*) of 6 m/s [23], Equations (1) and (2):

$$\text{Total travel time} = L/V = 6 \text{ m}/6 \text{ m/s} = 1 \text{ s} \tag{1}$$

$$\text{Time step size} = 1 \text{ s} \times (1/20) = 0.05 \text{ s} \tag{2}$$

A drawn-out computing time will be observed while using a time step size of 0.05 s (0.000, 835 min), which makes it unrealistic for a building analysis that could replicate long-term wind behavior (days and weeks) with the help of a Personal Computer (PC) (as illustrated in Table 1).

Table 1. Computing time required for 0.05 s time step size.

Wind Effect for One Day	Wind Effect for One Week	Wind Effect for 30 Days	Wind Effect for a Season (120 Days)
Simulation time around 180 days	Simulation time around 1200 days	Simulation time around 5350 days	Simulation time around 20,000 days

The simulation was performed with an Intel® Core™ i5-4200 U CPU@2.3 GHz with 8 GB RAM installed memory.

2.2. Divergence

For a typical solar heating analysis, the time step size can be comparable to 100 s or more. This will be able to determine divergence, when a bigger time step size (1 or 5 min) is taken into account for a long-term analysis. Once the additional iterations do not alter the outcome, then the divergence takes place. Since this process keeps on repeating till the time we observe an insignificant change in variable from a single iteration to the next one, in the event of deviated transient calculation, we will need to curtail the time step size, entailing a longer computing time [24].

Design of various models is intended for simulations with shorter duration (milliseconds, seconds, minutes, and hours, but not for months, weeks, or days); hence, very small steps are needed to implement the wind effect to the simulation. Accordingly, the simulation may achieve convergence or it results in excessive computing time. For instance, the analysis is terminated by the Autodesk Simulation CFD 2014 either when convergence is observed or upon completion of 750 iterations, whichever comes first [24].

It would not be possible to simulate the wind long term in the presence of smaller time steps. Therefore, to accelerate the simulation, a new technique was developed, where the wind effect will be combined with the external air temperature around the building. Previously, a number of different researchers have attempted to combine two different software programs for the long-term simulation of a building's thermal performance. Nevertheless, the time required for computation can be excessive. In the proposed technique, a single program is employed (CFD) with increased power for the purpose of simulating wind by including the effects of wind into the equivalent outside air temperature around the building. This enables larger time steps to be used (with shorter computing times) without divergence of the simulation.

CFD is a powerful simulation tool that can give great accurate results in the future of building thermal analysis if we could overcome the lengthy computing time to simulate the wind effect. If smaller time steps are used for wind, the long-term simulation would not be possible, which is the motivation behind the development of a new technique for accelerating the simulation by combining the wind effect with outside air temperature around the building. This issue was solved in this paper with the help of an equivalent temperature (known as $T_{natural}$), where the convection heat loss is

produced in an equal effect by the built-in wind effects to the outside air temperature around the building. Subsequently, this new external air temperature $T_{natural}$ can be used to calculate the internal air temperature for an extended period of time with a short CFD simulation time.

3. Primary Methods Adopted in This Research

3.1. Housing Test Modules

The validity of the assumptions integral to the new method was corroborated through the data acquired from four full-scale housing test modules. The Priority Research Center for Energy at the University of Newcastle, Australia has been engaged in a continuing research project focused on the thermal performance of houses for over a decade. Four full-scale housing modules were constructed within the scope of the research, and they are being monitored under various seasonal conditions, so that their thermal performance may be analyzed—see Figure 1 [25,26].
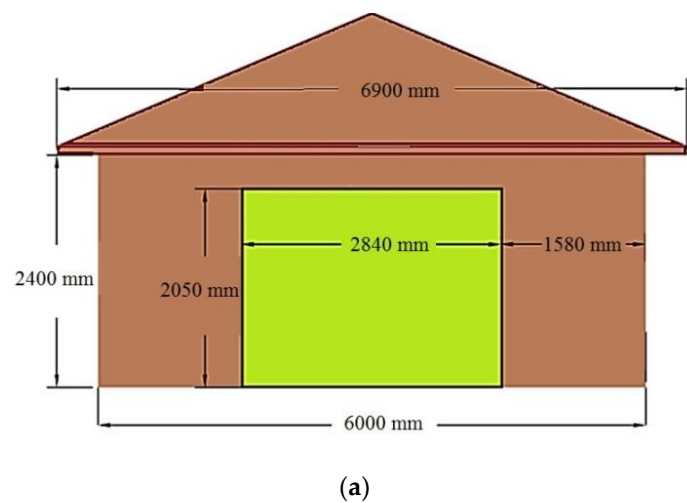

(a)

(b)

Figure 1. (**a**) Modules' layout, (**b**) actual site for the four test modules.

The housing test modules formed part of a wider research project at the University of Newcastle in Newcastle, Australia in which the housing test modules were included. Autodesk CDF simulation was employed for replicating each of the four full-scale housing modules. The simulation was characterized by the outside temperature acquired from the test modules.

All of the modules were developed at the Callaghan Campus of the University of Newcastle. The modules were representative of the standard types of building constructed in Australia. As illustrated in Figure 1, each of the modules was symmetrical, with a square floor plan of 6 m × 6 m and they were situated at a distance of 7 m from each other. This configuration was intended to avert shading and reduce any obstruction of the wind [27].

There were different walling systems of each module with the features given below [28–30]:

- An entry door; southern wall contains heavily insulated door to exclude any heat losses.
- Each northern wall contained a window of aluminum frame with 6.38 mm laminated clear glass.
- The concrete or clay roof tiles with sparking insulation were used for the roof. A 10 mm plasterboard ceiling was used with R-Values of 3.5 m^2·K/W (R3.5) glasswool batts insulation between rafters.
- The ground concrete floor incorporates a 100 mm thick concrete slab.

The walling system was the only difference between the modules:

- Cavity Brick Module (CB)
- There were two 110 mm brickwork skins in the walling system, where the internal walls were covered by 10 mm render and a 50 mm cavity was present between the walls.
- Insulated Cavity Brick Module (InsCB)
- Reportedly, there are two 110 mm brickwork skins in the walling system, where 10 mm internal render covers the internal walls and with R1 polystyrene insulation shields the 50 mm cavity.
- Insulated Brick Veneer Module (InsBV)
- The walling system comprised an outside 110 mm brickwork skin in addition to an inside timber frame with low-glare reflective foil as well as R1.5 glasswool batts, where the inside surface was covered by 10 mm plasterboard.
- Insulated Reverse Brick Veneer Module (InsRBV).

The walling system was basically comprised of a timber stud frame covered by R.15 glasswool batts insulation that was attached to an outside wall of 2–3 mm acrylic render on 7 mm fiber–cement sheets. The inside wall consisted of a 110 mm brick skin covered in 10 mm internal render [27].

In order for the inside air temperature and the outside environment to be determined, each of the modules had sensors installed, where at least 40 sensors were used in accordance with The American Society of Heating, Refrigerating and Air-Conditioning Engineers (ASHRAE55) for the purpose of measuring a small room's thermal comfort performance. The distribution of the sensors was designed so that they were situated in the walls and in the center of the module and not close to the occupied boundary, radiation, and diffusers (see Figures 2 and 3).
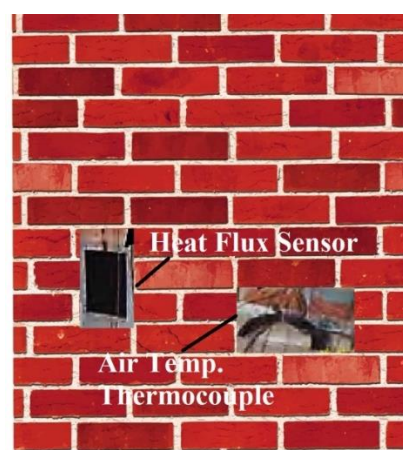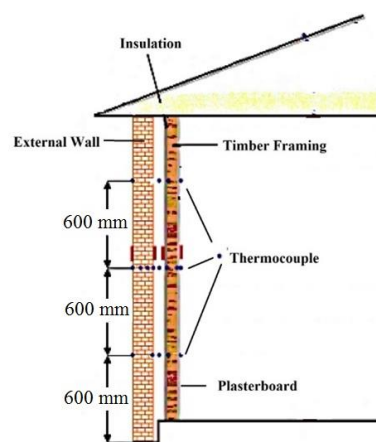

Figure 2. Internal wall sensors.

Figure 3. Sensors types and locations.

Datataker DT600 was used for recording the data at intervals of 5 min for 24 h/day throughout the testing period [27].

In this study, calculation of the operative temperature was performed by determining the average air temperature for the sensors situated in the center of the building at various heights: 600, 1200 and 1800 mm. Figure 3 shows a schematic of the standard sensor configurations for all of the types of wall. Modeling of the geometrical properties for each of the modules and their materials was performed with a CFD environment [28].

3.2. Computational Fluid Dynamics (CFD)

CFD was used to construct a large spherical-shaped outside environment with an external volume of 100 m × 100 m × 100 m. Subsequently, the material characteristics for every module were allocated with identical thermal properties to the actual modules. An automatic mesh was created for analyzing the modules utilizing an automatic topological examination for the overall geometry to determine the node distribution and size of the mesh. For the purpose of this analysis, a total of 264,534 nodes and K-epsilon turbulence were employed. Lastly, a grid independence test was performed to verify the accuracy of the CFD simulation. Figure 4 shows the specification of the temperature and emissivity boundary for the both the ground and sky.
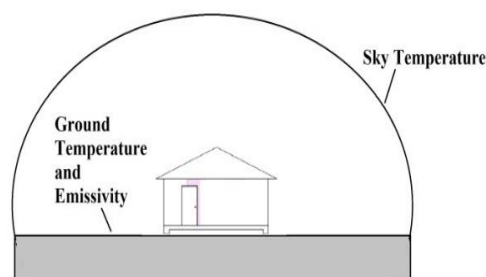

Figure 4. Sky temperature and ground emissivity used in the computational fluid dynamics (CFD) analysis.

In this study, Autodesk CFD 2014 was utilized, which is a simulation software used for the mathematical analysis of fluid flow and heat transfer based on numerical techniques and algorithms [14]. CFD analysis was used for precisely modeling the physical properties of each of the modules. After this, a grid independence test was performed to verify the precision of CFD simulation using data acquired from the actual modules.
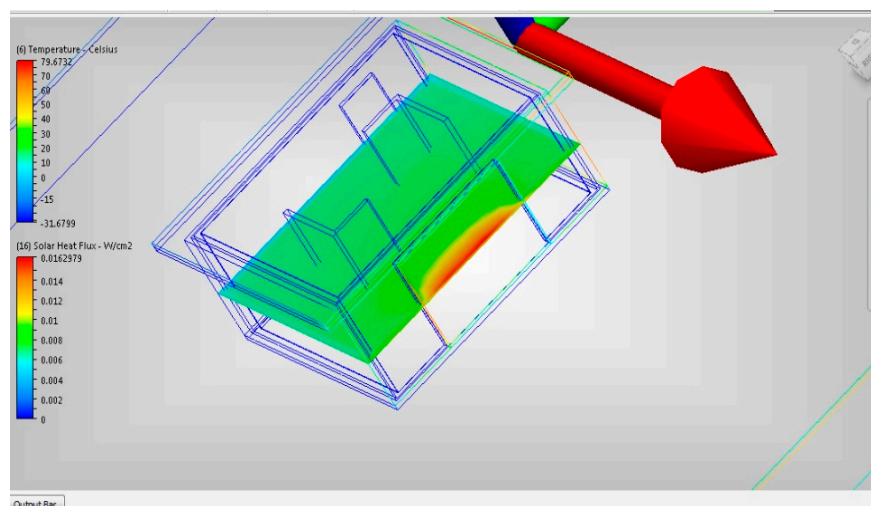
To find the internal air temperatures for all the modules, the average internal air volume temperatures were calculated using CFD simulations as shown in Figure 5.
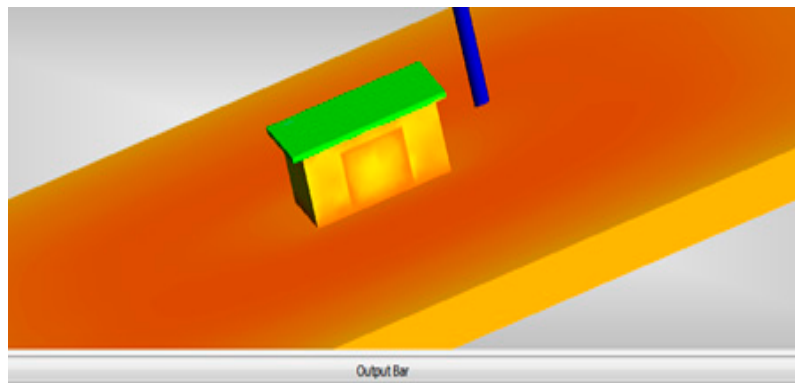
(**a**)
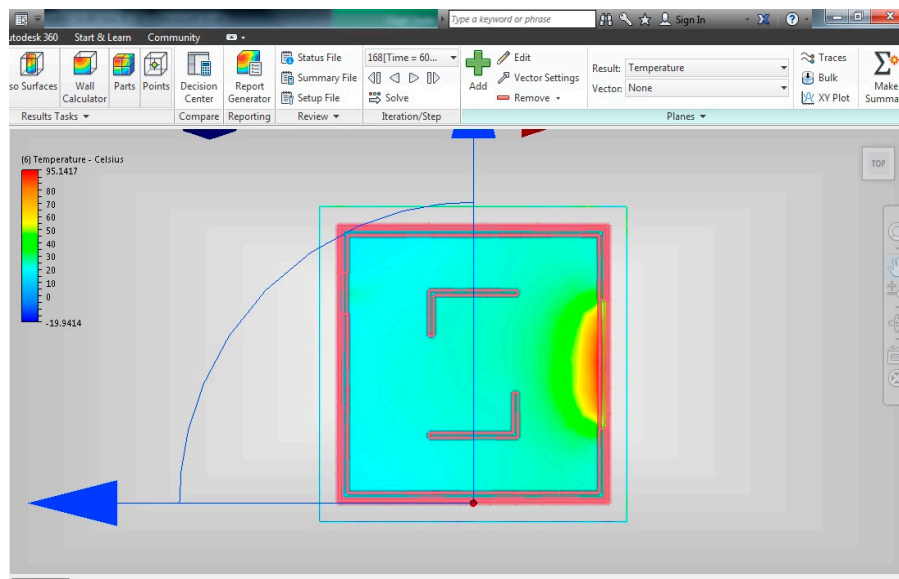
(**b**)

(**c**)

Figure 5. (**a**) Snapshot from the CFD simulation showing the external surfaces of the module. (**b**) Horizontal plane inside the insulated cavity brick (InsCB) module during midday in winter on the floor (elevation = 0). (**c**) Horizontal plane inside the InsCB module during midday in winter for 1200 mm elevation.

Note, we used the heavy insulted roof to neglect the heat transfer between the roof and the internal surfaces.

3.3. Wind Data at Site

The direction and speed of the wind tends to change with time. Figures 6 and 7 show the speed and direction of the wind recorded at intervals of 5 min at the location for a duration of a week between 14 January 2020 and 21 January 2020.
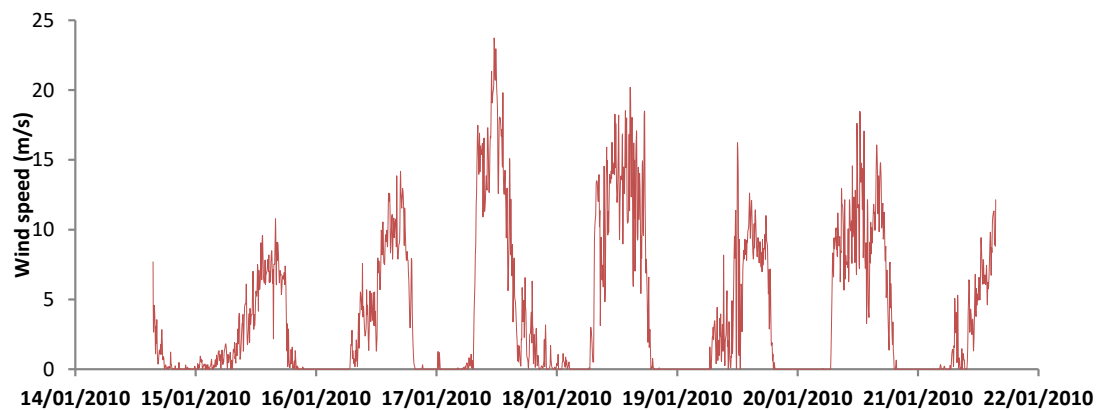

Figure 6. Depicts the wind speed recorded at the top of the building for one week (4 m from the ground level).
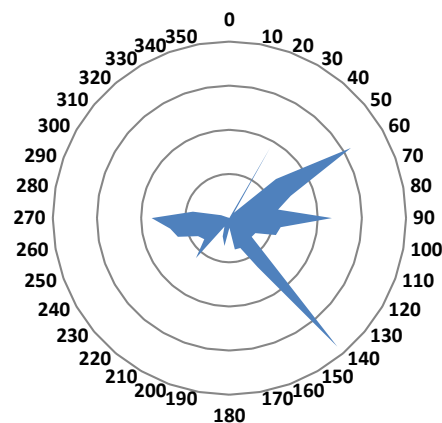

Figure 7. The direction of wind and its frequency (calculation of time if wind from each direction).

Mostly, East/North and South/East are the key sources of wind direction. The western side is known for its calm wind, as demonstrated in Figure 7.

4. Methodology

Previously, a number of different researchers have attempted to combine two different software programs for the long-term simulation of a building's thermal performance. Nevertheless, the time required for computation can be lengthy. In the proposed technique, a single program is employed for the purpose of simulating wind by including the effects of wind into the equivalent outside air temperature around the building. This enables larger time steps to be used (with shorter computing times) without divergence of the simulation [14].

To calculate the new external air temperature that triggers an identical volume of convection heat losses to that of wind, the following equation of convective heat transfer was utilized [29], Equation (3)

$$q = h_c A_s \ (T_s - T_{air}) \tag{3}$$

where
　　A_s—surface area (m^2).
　　q—heat energy (W).
　　T_{air}—ambient air temperature (K).
　　T_s—surface temperature (K).
　　h_c—convective heat transfer coefficient ($W/m^2 \cdot K$).

The wind speed is likely to change the heat transfer coefficient. To calculate the air temperature $T_{natural}$ (with no wind effects) that triggers the same rate of convection heat loss as T_{air} (with wind effects), we can write the equation as follows, Equation (4):

$$q = h_{natural}\,A_s\,(T_s - T_{natural}) = h_{total}\,A_s\,(T_s - T_{air}) \tag{4}$$

After simplification, we get, Equation (5):

$$T_{natural} = T_s - (h_{tota}/h_{natural})(T_s - T_{air}) \tag{5}$$

T_{air}—outside air temperature with wind speed = V_{actual}
$T_{natural}$—outside air temperature with wind speed equal zero (no wind effect)
h_{total}—total heat transfer coefficient (forced and natural exterior convective coefficient)
$h_{natural}$—natural convection heat transfer coefficient with no wind = 3.5 $W/m^2 \cdot K$ [30].

In the design stage, the outside surface of a multi-layer wall subjected to diurnal fluctuations in the temperature of the external environment due to solar heating in the daytime and the sky temperatures at night-time were theoretically calculated. The measured temperature of the external surface is correlated with the theoretical equivalent sol-air temperature as well as the internal and external air temperatures via the numerical solution of the heat equation that is utilized for estimating the heat flux through a non-homogenous medium [31].

The convective heat transfer coefficients are generally used to simulate the convective heat exchange at an external building surface owing to air flow along the surface h_{forced}, and the same can be calculated as follows [32], Equations (6) and (7):

$$h_{forced} = 5.01(U_{10})^{0.85} \text{ for WW} \tag{6}$$

$$h_{forced} = 2.27(U_{10})^{0.85} \text{ for LW} \tag{7}$$

where
　　U_{10}—wind speed at a height of 1 m above the ground, intended for anemometers in weather stations.
　　h_{forced}—for the outside surfaces of buildings, forced convective heat transfer coefficients.
　　WW—windward incidence angles in the range between −90 and 90.
　　LW—leeward (remaining incidence angles), where the incidence angle is defined as the angle between the standard to the windward surface and the wind direction of the approach flow.

The quadratic sum of the forced and natural convection elements can provide an indication of the exterior total convection heat transfer coefficient [33], Equation (8):

$$h_{total} = [h^2_{force} + h^2_{natural}]^{0.5} \tag{8}$$

where
　　h_{forced}—forced exterior convective coefficient.
　　$h_{natural}$—natural exterior convective coefficient.
　　h_{total}—total (forced + natural) exterior convective coefficient.

The wind data was recorded at a height of approximately 4 m from the ground (z_r), with the aim of computing the related speed of the wind at a height of 10 m (z). For this purpose, they utilized the logarithmic law [34], Equation (9):

$$\frac{u(Z)}{u(Z_r)} = \frac{\ln\left(\frac{Z}{Z_o}\right)}{\ln\left(\frac{Z_r}{Z_o}\right)} \tag{9}$$

where

$u(Z)$—wind speed at a height of 10 m

Z_o—roughness of the surface (few trees $z = 0.1$ m [34]). The terrain of the University of Newcastle, Australia.

$u(Z_r)$—recorded wind speed at a height of 4 m (the height of the anemometer above the building). For a 10 m height, $U_{10} = 1.661 \times U_4$ (at 4 m height).

Note: The speed at a height of 10 m and the wind speed detected at the top of the modules multiplied by 1.661 are the same (i.e., the position of the anemometer).

By implementing the equation shown above on the new outside layer surrounding the building (see Figure 8), it was possible to analyze the modules using the CFD for an extended time. This was achieved by including the effect of the wind into the outside air temperature close to the building.
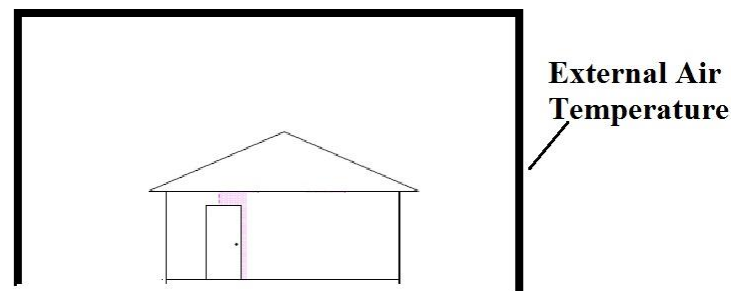

Figure 8. New external air layer used to include wind effect in the CFD simulations.

The analysis was conducted for every module utilizing CFD incorporating the external air temperature air temperature, $T_{natural}$, which includes the effects of the wind in addition to initiating the identical convection heat loss rate to T_{air} (with wind effect). Actual data were then assessed in order to make a comparison with the results of the simulation [14].

5. Results and Discussion

A comparison was made between simulations conducted with this technique and actual data from four extant housing test modules constructed from a variety of walling systems, which produced a precise, representative analysis along with a significant reduction in the time required for the simulation. For each module's outside wall surfaces (North, South, East, and West, apart from the roof as it had extensive insulation to prevent all heat exchange), the new outside air temperatures including the wind effect were implemented in the new layer around the modules. This procedure was applied to all the modules. The Figures 9–12 show an example for one module (InsRBV).

Higher wind speeds increase the wind effect, which results in an increase in the new external air temperature particularly with the wind effect on the windward wall. Since most wind came from the East and West directions (see Figure 3), the effect on the eastern and western wall was much higher compared to that on the northern and southern walls as depicted in Figures 11 and 12.

From the above, if the wall surface temperature is higher than the external air temperature, the new external air temperature with the wind effect will be lower than the wall surface temperature, and therefore the wind will cool the wall surfaces. Conversely, if the wall surface temperature is lower than the external air temperature, the new external air temperature with wind effect will be higher than the wall surface temperature, with the wind heating the wall surfaces.
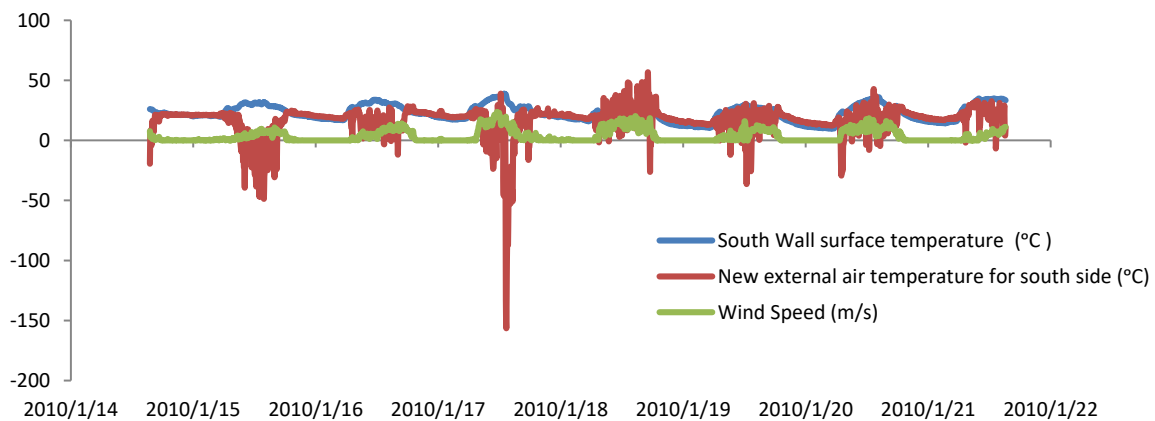
Figure 9. Wall temperature, new air layer temperature, and wind speed for the southern wall.
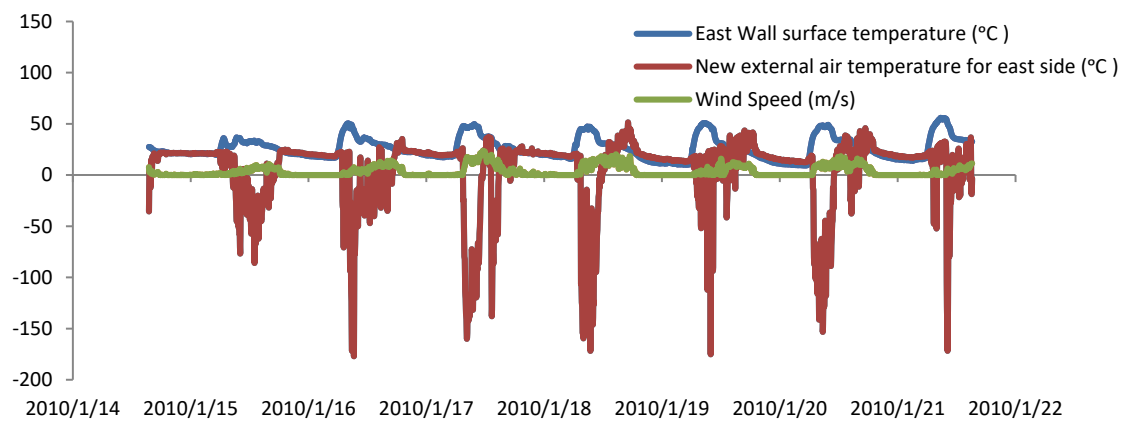

Figure 10. Wall temperature, new air layer temperature, and wind speed for the eastern wall.
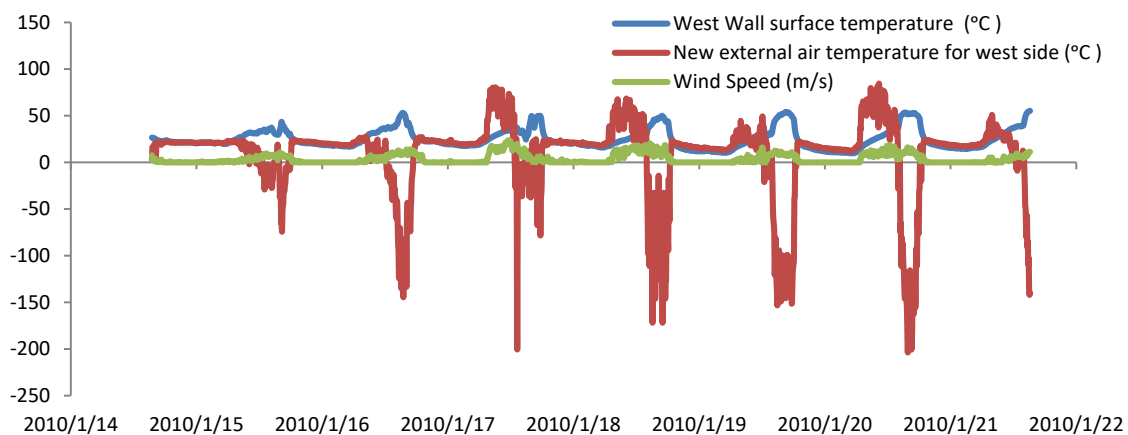

Figure 11. Wall temperature, new air layer temperature, and wind speed for the western wall.

By applying the above approach with the wind effects included in the external air layer surrounding the building, the internal temperatures shown in Figure 12 were obtained for the CFD simulation. Comparisons with the observed (real) data are shown for all modules using an 80 min time step size when the above approach is applied with the wind effects included in the external air, then the CFD simulation assists us in getting the internal temperatures as shown in Figure 13.

By using an 80 min time step size, the given image depicts the comparisons with the observed (real) data for all modules [12].
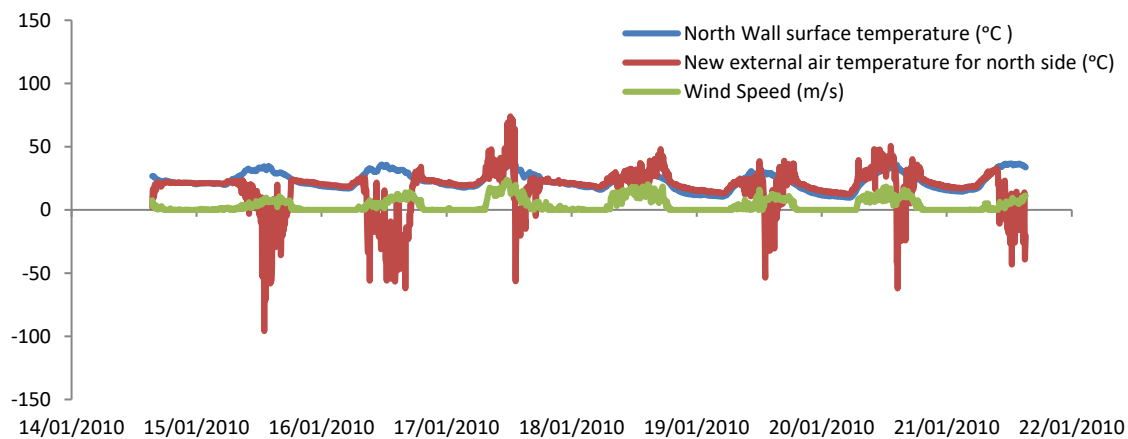
Figure 12. Wall temperature, new air layer temperature, and wind speed for the northern wall.

A comparison was made between the CFD simulation and the actual inside air temperature in terms of the number of hours that each of the modules spent within the temperature difference (range), as demonstrated in Table 2.

Table 2. Percentage for the number of hours for every temperature difference between actual data and the CFD simulation for the period studied.

Temperature Range (°C)	Insulated Reverse Brick Veneer Module	Cavity Brick Module	Insulated Brick Veneer Module	Insulated Cavity Brick Module
0–1	31%	35%	41%	52%
0–2	73%	65%	81%	83%
0–3	94%	92%	95%	98%
0–4	100%	98%	100%	100%

Percentage error is the difference between the recorded data on the site (real values) and the simulated values, divided by the real values, multiplied by 100 to give a percentage.

It is evident that the application of wind to the outside later around the modules will facilitate a more precise calculation of the inside air temperature, whereby in excess of 90% of the simulated results were in the range of 0–3 °C in comparison to the actual data (91% for CB, 99% for InsCB, 94% for InsBV, 93% for InsRBV modules).

The use of the new $T_{natural}$ will produce results with reduced accuracy in comparison to those generated with the inclusion of wind effects. Table 3 shows that when the wind effects were ignored, the results became less accurate.

Table 3. Percentage of the number of hours for every temperature difference between actual data and the CFD simulation for the period studied (utilizing new $T_{natural}$).

Temperature Range (°C)	Insulated Reverse Brick Veneer Module	Insulated Cavity Brick Module	Insulated Brick Veneer Module	Cavity Brick Module
0–1	32%	47%	41%	32%
0–2	68%	77%	76%	62%
0–3	84%	89%	87%	88%
0–4	99%	97%	98%	96%

Upon inclusion of wind effects, above 90% of the results were found within 0–3 °C of the perceived temperatures. However, a decline of 83–88% was observed in the results after ignoring the wind effects. Hence, presence of wind effects holds greater importance in correct simulation of the thermal performance of the modules.
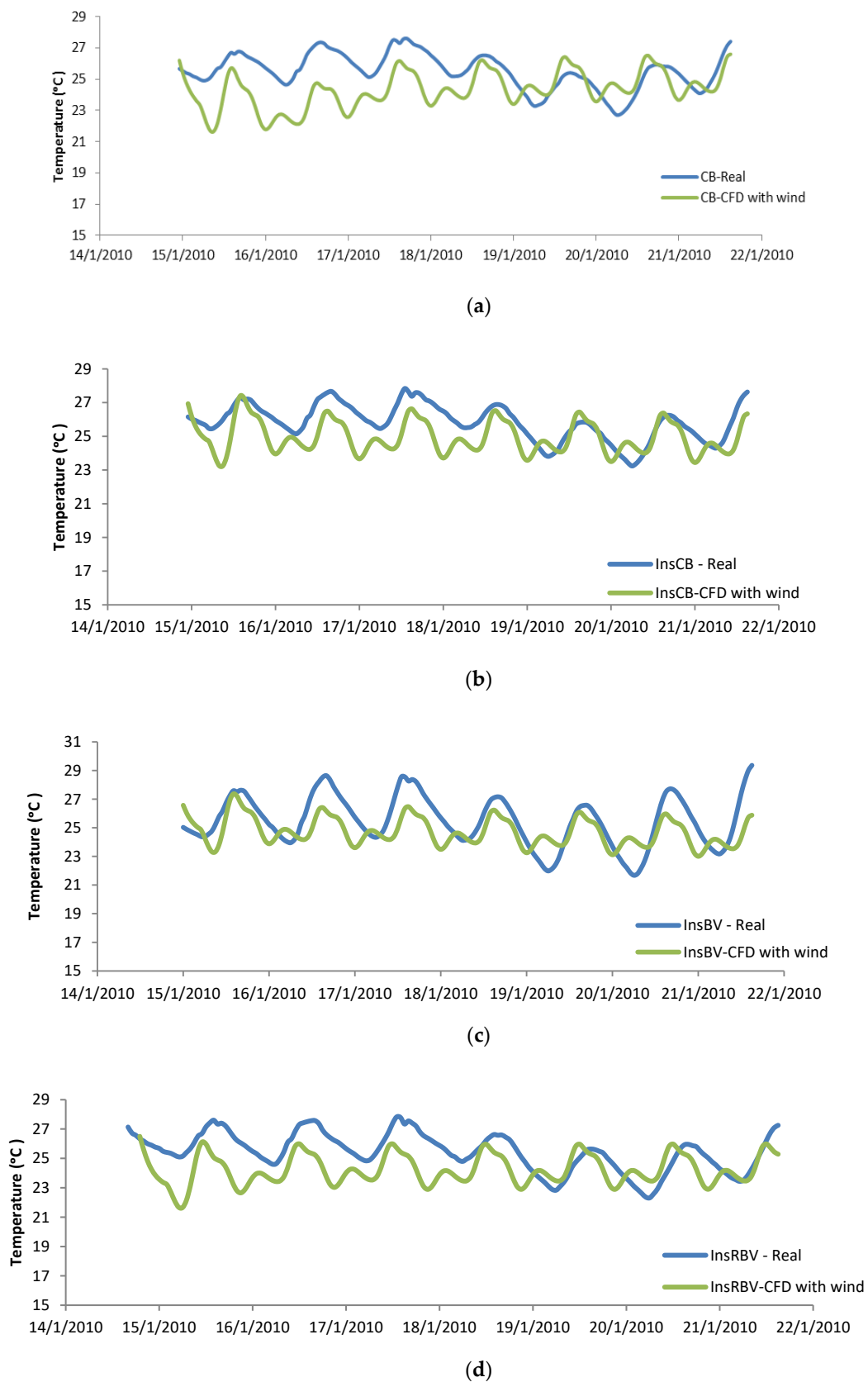
Figure 13. Building internal air temperature for CFD simulation with wind effect and real data (**a**) cavity brick (CB) module, (**b**) InsCB module, (**c**) insulated brick veneer (InsBV) module, (**d**) insulated reverse brick veneer (InsRBV) module.

The use of a larger time step is enabled by directing wind effect to the external air temperature layer, for example, 60, 80, or 100 min time step size. Moreover, shorter computing times would be required by the CFD simulation process, as shown in Table 4.

Table 4. Simulation time for extended time step sizes.

Time Step Size (Minutes)	Simulation Duration for One Week	Simulation Duration for 30 Days	Simulation Duration for 120 Days
Sixty	29 min.	1 h 50 min.	7 h 15 min.
Eighty	24 min.	1 h 20 min.	5 h 25 min.
One hundred	19 min.	1 h 10 min.	4 h 25 min.

The computing time becomes curtailed and the use of CFD simulations becomes easy if wind effect is incorporated into the external air temperature. Consequently, it helps researchers to examine building performance over long periods. According to the previously mentioned new technique, the wind effect surrounding the buildings is incorporated into the outside air temperature by using a matching temperature (defined in this case as $T_{natural}$), which enables the identical rate of convection heat loss to be generated as it was generated with the wind effects. Moreover, this new external air temperature, $T_{natural}$, will be used to compute the internal air temperature of the building.

Almost 90% of the simulated CFD results remained within 0–3 °C for the one-week analysis on a desktop PC. Verification of the modeling approach was achieved by comparing the inside air temperatures simulated by CFD with the real temperatures recorded in each of the full-scale housing modules, each of which had a distinct walling system. The average accuracy generated by this comparison was greater than 90% for each of the simulated modules for any given time during the 12-month simulation period. Such results were in accordance with both the assessment software employed in Australia (i.e., AccuRate; official building rating tool) as well as previous findings from studies on walling systems conducted at the University of Newcastle in Australia [25].

6. Conclusions

The heating and cooling loads in thermal design are directly affected by the internal air temperature of the building. While considering the wind effects, very small time steps (seconds and milliseconds) are required for simulation of the thermal performance of buildings. Multiple previous studies have attempted to simulate long-term wind effects by combing two different software packages to simulate the thermal performance of buildings for an extended period. Nevertheless, the computation process involved can be lengthy. In the proposed technique, a single program (CFD) with increased power is used for the simulation of wind by including wind effects into the corresponding outside air temperature around the building. This enables larger time steps to be used (with shorter computing times) with no divergence of the simulation. Therefore, CFD analysis via personal computers becomes inappropriate for long-term building analysis (i.e., weeks, months), especially when the process will demand unrealistically long computing times (approximately years). In this research, a method was presented for bringing accurate results and faster simulation. As far as this method is concerned, the external air temperature adjoining the building basically incorporated the wind effect, and larger time steps (minimizing computing time) were utilized to perform the analysis without any deviation in the simulation results.

CFD is a powerful simulation tool that can give great accurate results for building thermal analysis if we could overcome the lengthy computing time to simulate the wind effect. Hence, it is possible to calculate the inside air temperature of the building through the adoption of a new outside air temperature, $T_{natural}$, which considers the identical rate of convection heat losses generated by the wind around the structure. Thus, $T_{natural}$ allows the wind effect to be directly included and leads to a significant decrease in the total computation time. This study has presented the results of simulations conducted in January (southern hemisphere summer) and then a comparison was made with actual

data acquired from housing test modules located on the campus of the University of Newcastle. The findings revealed that a significant reduction in the simulation time was achieved while the rate of accuracy of over 90% was maintained; greater than 90% of the simulated data were in the 0–3 °C range in comparison to the actual data. The proposed method (which can be implemented for all types of buildings with high accuracy) led to a reduction in time required for computation for the one-week analysis using a desktop PC from in excess of 3 years to under 30 min, which demonstrates that it is highly effective.

Author Contributions: Conceptualization, D.A.; Formal analysis, A.A.; Investigation, A.A. Project administration, B.M.; Supervision, A.P. and B.M.; Writing—original draft, A.A.; Writing—review & editing, A.P. All authors have read and agreed to the published version of the manuscript.

Conflicts of Interest: The authors declare no conflict of interest.

References

1. Al-Addous, M.; Albatayneh, A. Knowledge gap with the existing building energy assessment systems. *Energy Explor. Exploit.* **2019**, *38*, 783–794. [CrossRef]
2. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. The significance of sky temperature in the assessment of the thermal performance of buildings. *Appl. Sci.* **2020**, *10*, 8057. [CrossRef]
3. Chang, H.; Xiang, C.; Duan, C.; Wan, Z.; Liu, Y.; Zheng, Y.; Shang, Y.; Liu, M.; Shu, S. Study on the thermal performance and wind environment in a residential community. *Int. J. Hydrog. Energy* **2016**, *41*, 15868–15878. [CrossRef]
4. De Oliveira Neves, L.; da Silva, F.M. Simulation and measurements of wind interference on a solar chimney performance. *J. Wind Eng. Ind. Aerodyn.* **2018**, *179*, 135–145. [CrossRef]
5. Dodoo, A.; Tettey, U.Y.A.; Gustavsson, L. Influence of simulation assumptions and input parameters on energy balance calculations of residential buildings. *Energy* **2017**, *120*, 718–730. [CrossRef]
6. Hosseini, S.H.; Shokry, E.; Hosseini, A.A.; Ahmadi, G.; Calautit, J.K. Evaluation of airflow and thermal comfort in buildings ventilated with wind catchers: Simulation of conditions in Yazd City, Iran. *Energy Sustain. Dev.* **2016**, *35*, 7–24. [CrossRef]
7. Du, Y.; Mak, C.M.; Liu, J.; Xia, Q.; Niu, J.; Kwok, K.C. Effects of lift-up design on pedestrian level wind comfort in different building configurations under three wind directions. *Build. Environ.* **2017**, *117*, 84–99. [CrossRef]
8. Zhai, Z. Apply CFD to Design Comfortable and Energy-Efficient Buildings. In Proceedings of the ASME Pressure Vessels and Piping Conference, San Diego, CA, USA, 25–29 July 2004; Volume 46865, pp. 1–12.
9. Sousa, J. Energy simulation software for buildings: Review and comparison. In Proceedings of the International Workshop on Information Technology for Energy Applications-IT4Energy, Lisbon, Portugal, 6–7 September 2012.
10. Albatayneh, A.; Alterman, D.; Page, A. Adaptation the use of CFD modelling for building thermal simulation. In Proceedings of the 2018 ACM International Conference on Software Engineering and Information Management, Casablanca, Morocco, 3–5 January 2018; pp. 68–72.
11. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. Discrepancies in peak temperature times using prolonged CFD simulations of housing thermal performance. *Energy Procedia* **2017**, *115*, 253–264. [CrossRef]
12. Albatayneh, A.; Alterman, D.; Page, A.W.; Moghtaderi, B. Warming issues associated with the long term simulation of housing using CFD analysis. *J. Green Build.* **2016**, *11*, 57–74. [CrossRef]
13. Albatayneh, A.; Alterman, A.; Page, A.; Moghtaderi, B. The significance of time step size in simulating the thermal performance of buildings. *Adv. Res.* **2015**, *5*, 1–12. [CrossRef]
14. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. An alternative approach to the simulation of wind effects on the thermal performance of buildings. *Int. J. Comput. Phys. Ser.* **2018**, *1*, 35–44. [CrossRef]
15. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. The significance of building design for the climate. *Environ. Clim. Technol.* **2018**, *22*, 165–178. [CrossRef]
16. Crawley, D.B.; Pedersen, C.O.; Lawrie, L.K.; Winkelmann, F.C. EnergyPlus: Energy simulation program. *ASHRAE J.* **2000**, *42*, 49.

17. Zhai, Z.J.; Chen, Q.Y. Sensitivity analysis and application guides for integrated building energy and CFD simulation. *Energy Build.* **2006**, *38*, 1060–1068. [CrossRef]
18. Yi, Y.K.; Feng, N. Dynamic integration between building energy simulation (BES) and computational fluid dynamics (CFD) simulation for building exterior surface. In *Building Simulation*; Tsinghua University Press: Beijing, China, 2013; Volume 6, pp. 297–308.
19. Wang, L.; Wong, N.H. Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment. *Build. Environ.* **2009**, *44*, 95–112. [CrossRef]
20. Zhang, Z.; Zhang, W.; Zhai, Z.J.; Chen, Q.Y. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part 2—Comparison with experimental data from literature. *HVAC R Res.* **2007**, *13*, 871–886. [CrossRef]
21. Zhai, Z. Application of computational fluid dynamics in building design: Aspects and trends. *Indoor Built Environ.* **2006**, *15*, 305–313. [CrossRef]
22. Autodesk Simulation, CFD, 2014. Solar Heating. Available online: http://help.autodesk.com/view/SCDSE/2014/ENU/?guid=GUID-EBC7C1DC-11F8-4BC2-8D0D-D49AB3106848 (accessed on 1 November 2020).
23. Autodesk Simulation, CFD, 2014. Transient Flows. Available online: http://help.autodesk.com/view/SCDSE/2014/ENU/?guid=GUID-384EC529-309C-426C-B036-93ADDB842F0E (accessed on 1 November 2020).
24. Autodesk Simulation, CFD, 2014. Transient Parameters. Available online: http://help.autodesk.com/view/SCDSE/2014/ENU/?guid=GUID-7BE54C33-D7CE-4F95-81B0-FD3B5517E9B3 (accessed on 1 November 2020).
25. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. Development of a new metric to characterise the buildings thermal performance in a temperate climate. *Energy Sustain. Dev.* **2019**, *51*, 1–12. [CrossRef]
26. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. The significance of the adaptive thermal comfort limits on the air-conditioning loads in a temperate climate. *Sustainability* **2019**, *11*, 328. [CrossRef]
27. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. The impact of the thermal comfort models on the prediction of building energy consumption. *Sustainability* **2018**, *10*, 3609. [CrossRef]
28. Albatayneh, A.; Alterman, D.; Page, A.; Moghtaderi, B. The significance of temperature based approach over the energy based approaches in the buildings thermal assessment. *Environ. Clim. Technol.* **2017**, *19*, 39–50. [CrossRef]
29. Cengel, Y.A. *Heat and Mass Transfer: A Practical Approach*; McGraw-Hill: New York, NY, USA, 2006.
30. Luo, C.; Moghtaderi, B.; Hands, S.; Page, A. Determining the thermal capacitance, conductivity and the convective heat transfer coefficient of a brick wall by annually monitored temperatures and total heat fluxes. *Energy Build.* **2011**, *43*, 379–385. [CrossRef]
31. Marino, B.M.; Muñoz, N.; Thomas, L.P. Calculation of the external surface temperature of a multi-layer wall considering solar radiation effects. *Energy Build.* **2018**, *174*, 452–463. [CrossRef]
32. Defraeye, T.; Blocken, B.; Carmeliet, J. Convective heat transfer coefficients for exterior building surfaces: Existing correlations and CFD modelling. *Energy Convers. Manag.* **2011**, *52*, 512–522. [CrossRef]
33. Booten, C.; Kruis, N.; Christensen, C. *Identifying and Resolving Issues in EnergyPlus and DOE-2 Window Heat Transfer Calculations (No. NREL/TP-5500-55787)*; National Renewable Energy Laboratory (NREL): Golden, CO, USA, 2012.
34. Manwell, J.F.; McGowan, J.G.; Rogers, A.L. *Wind Energy Explained: Theory, Design and Application*; John Wiley & Sons: Hoboken, NJ, USA, 2010.