INVESTIGATION OF THE POTENTIAL BENEFITS OF OPTIMIZING BUILDING ELEMENT PLACEMENT USING COMPUTATIONAL FLUID DYNAMICS

Nastaran Shahmansouri [1], Rhys Goldstein [1], Farhad Javid [1], Alex Tessier [1], Simon Breslav [1], Azam Khan [2]

[1]Autodesk Research, Toronto, Canada
[2] Department of Computer Science, University of Toronto, Canada

ABSTRACT

Buildings are responsible for more than one-third of global energy consumption, motivating the use of modeling to improve energy efficiency while maintaining occupant comfort. While conventional energy models are based on well-mixed zones, we explore the potential benefits of using high-fidelity models to optimize the exact placement of building elements. Specifically, we model a single-room environment and apply computational fluid dynamics to compare 36 mechanical configurations of supply and return vents combined with 9 occupant locations. Results indicate the same forced-air cooling input may produce location-specific temperatures and air velocities that vary significantly depending on the configuration of building elements.

INTRODUCTION

Building construction and maintenance are responsible for more than one-third of global energy consumption and generate, directly and indirectly, nearly 40% of total CO_2 emissions (Al Horr et al. 2017; Sinha, Lennartsson, and Frostell 2016). Moreover, it has been shown that indoor environmental conditions and thermal comfort can hugely impact the health, well-being, and productivity of occupants (ASHRAE 2009; Al Horr et al. 2016; Mendell et al. 2002). Energy modeling receives considerable attention as a source of insight toward improving the energy efficiency of buildings and the comfort of the people who occupy them, though the impact and potential of current methods is open to debate (Mahdavi 2020).

The conventional practice for energy modeling is to use simplified energy models, where a uniform temperature is predicted for each zone (Kato 2018). Most building energy modeling tools are based on the well-mixed zone air assumption, under which the exact placement of building elements such as supply and return vents (i.e. the mechanical configuration), windows and room partitions (i.e. the architectural configuration), and furniture (i.e. occupant locations) have little impact on energy efficiency and occupant comfort. Many of these elements are simplified or neglected in conventional models (Kim et al. 2015; Lee 2007). Yet depending on the mechanical, occupant, and architectural configurations, the well-mixed assumption may be not sufficient. A design based on conventional building energy modeling tools may conceal the potential severity of a poor overall configuration of elements.

Advancements in simulation techniques and computing power provide a means to employ more detailed models in building design processes. An alternative to simplified energy modeling with the well-mixed zone air assumption is to combine building energy network modeling and computational fluid dynamics (CFD) simulations (Kato 2018). By integrating high-fidelity CFD simulations into energy modeling practice, airflow and thermal comfort could be predicted for various positions and times in space. CFD analysis has the potential to help building design and engineering professionals understand flow properties over a refined temporal and spatial grid for different mechanical and architectural configurations. An outstanding question is, how much improvement in energy use and comfort could potentially be gained by shifting toward such high-fidelity models?

The overarching objective of this research is to investigate the potential benefits of employing high-fidelity air flow simulations to plan the exact configuration of architectural and mechanical elements as well as the locations of occupants in an indoor environment. Building upon previous work, numerical techniques are employed to evaluate thermal comfort, specifically temperature and velocity variations, in localized volumes where people mostly occupy. The study is performed on a simplified indoor environment consisting of a room with cooling, ventilation, and window elements. The locations of the supply cooling vent (called supply) and return vent (called return) are systematically varied to cover multiple location on opposite walls of the space. Transient turbulent CFD analyses are conducted on a total of 36 supply-return configurations, see Figure 1. Predicted temperatures and airflow velocities are then sampled at 9 locations representing different possible locations of occupants.

It should be mentioned that this study is envisioned as an early step in a larger effort toward understanding (1) the degree to which spatial configurations affect comfort and energy usage in buildings, and (2) the potential benefits of incorporating high-fidelity CFD simulation into energy modeling practice in order to optimize element placement. Future studies of a similar nature could look at environments with multiple rooms and circulation areas, multiple occupants with locations determined by furniture arrangement, multiple supply/return vents, and other complicating factors.

RELATED WORK

There has been increasing amount of interest in using CFD analysis in the field of building design (Kato 2018; Zhai 2006). The applications of CFD include site planning, natural ventilation studies, pollution dispersion and control, the prediction of fire and smoke movement in a building (Zhai 2006). Additionally, the integration of CFD and building energy modeling has attracted attention since it can provide more accurate information about indoor air quality and energy simulation. CFD has been used for heat transfer analysis in large areas such as atria and concert halls as well as smaller areas such a bedroom (Gilani, Montazeri, and Blocken 2016; Hussain and Oosthuizen 2012; Moosavi et al. 2014; Perén et al. 2015; Sakai et al. 2008; Stavridou and Prinos 2017; Z. J. Zhai and Chen 2005). Many studies have focused on natural ventilation and some studies have examined the effect of forced-air ventilation. Building CFD modeling efforts can be classified into two main categories: long term (for several month of a year) and short term (for less than an hour) (Kato 2018).

CFD modeling for room environments have been investigated for different applications and purposes. A number of studies have employed transient and steady-state analyses to understand the natural ventilations with heat sources and compared the results with experimental data (Al-Sanea, Zedan, and Al-Harbi 2012; Gilani, Montazeri, and Blocken 2016; Stavridou and Prinos 2017; Yang et al. 2015). These studies provide information on the effectiveness of CFD simulations and how to improve the accuracy of simulations for particular applications. Moreover, studies have been done on the optimization of indoor air conditioning with active (HVAC) and passive design elements using CFD and provided suggestions in this area (Lee 2007). Studies have been conducted on the methods to employ CFD modeling for building control applications (Kim et al. 2015). Research has been done on estimating the allowable air return of an HVAC system that minimizes energy costs while controlling indoor air quality (Kanaan 2019). Numerical investigations have been conducted on the impact of exhaust height on energy saving and indoor air quality for a room with a workstation (Ahmed and Shian 2017).

This study, by contrast, aims to investigate the impact of the placement of elements of a mechanical system in conjunction with elements that are conventionally considered at different stages of the design process. It is envisioned as a step toward understanding the full impact of detailed building geometry on air flow patterns and ultimately building performance. If the impact is low, then CFD should arguably continue to be limited to the specific engineering use cases where it is currently applied. If the impact is high, then future investigation is needed to determine the benefit and viability of radically expanding the use of high-fidelity simulations. Considering the computational costs of high-fidelity models, surrogate modeling may make it more feasible to apply CFD at larger scales than previously possible. It may allow high-fidelity building simulations and multi-level optimization to be expanded to include multiple connected indoor spaces as well as HVAC systems (Gorissen, Dhaene, and De Turck 2009).

SIMULATION

In this paper, the *mechanical configuration* refers to the simplified building with particular combined placements of supply and return. The *occupant location* refers to the locations where the velocity and temperature data have been evaluated. These locations are assumed to be representative of small localized volumes occupied most of the time by an occupant. The *combined configuration* refers to a particular location where the velocity and temperature data have been evaluated in the simplified building with a specific supply and return configuration.

The simplified building modeled has a forced-air cooling system, return ventilation, and a window. A wide range of element placements are tested to investigate the degree to which spatial configurations affect comfort and energy use.

Configurations

The modeled building has a width of 4.8 m, a depth of 4 m, and a height of 2.7m. The supply and return are located in two opposite walls, as shown in Figure 1. The returns are located in the wall at $z = 0$, and the supplies are placed in the wall at $z = 4$ m. The supplies and returns are rectangle vents with a fixed width and height of 0.75 m and 0.5 m, respectively. The supply and return of each configuration are placed in one of the six locations specified in Table 1. This table reports the locations of the corner with minimum x and y values.

Table 2 tabulates the information for all 36 mechanical configurations (6 supply × 6 return) and associates each configuration with an ID. The size and location of the window are kept fixed; the corner of the window with minimum values is placed at (x = 0, y = 0.6, z = 1). The window has a height of 1.5 m and a depth of 2 m. The velocity and temperature data were obtained and analyzed for 9 locations, i.e. the occupant locations, shown in blue in Figure 1. Table 3 presents the position and ID of these locations. The data was obtained for a height of 1.7 m, representing the average standing height of a person.

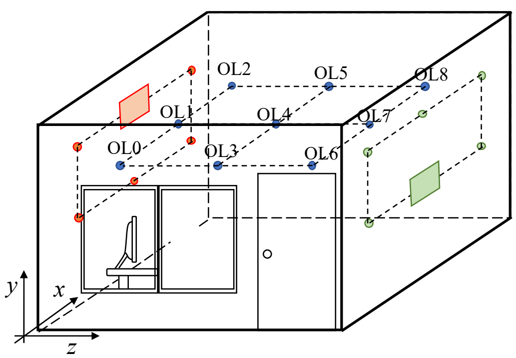


Figure 1 Schematic of the simplified building. The center-point of all 6 supply and 6 return locations are respectively shown in green and red. Temperature and velocity data are obtained for 9 locations inside the room at a height of 1.7m shown with blue markers.

Table 1 The locations of the supply and return. The values show the corner with minimum (x, y) values.

SUPPLY (X, Y, Z)	RETURN (X, Y, Z)
(0.425,0.425,4)	(0.425,0.425,0)
(0.425,1.775,4)	(0.425,1.775,0)
(2.025,0.425,4)	(2.025,0.425,0)
(2.025,1.775,4)	(2.025,1.775,0)
(3.625,0.425,4)	(3.625,0.425,0)
(3.625,1.775,4)	(3.625,1.775,0)

Computational fluid dynamics simulation

The motion of air in a room is described by the well-known Navier-Stokes equations, derived by applying the conservation laws of mass and momentum to a viscous fluid. The temperature is obtained from the conservation of energy equation. Navier-Stokes and energy equations can be coupled to determine the velocity, pressure, and temperature of a flow through time in a spatial domain. In Eulerian description, there is a nonlinear advection term in both momentum and energy equations which shows the transport of momentum and energy quantities due to the bulk motion (velocity) of the fluid. Due to the nonlinearity of this term, the time-dependent behavior of a fluid can be chaotic, e.g. turbulent. The turbulent behavior is observed when the ratio of the inertial energy to the dissipated viscous energy (defined by Reynolds number) is > 4,000. Considering the common properties of HVAC and the buildings, the ventilation flow inside a building is usually turbulent (Kato 2018).

Table 2 Simulation cases and their representative IDs for architectural configurations: Supply and return values show the corner with minimum (x, y) values.

ID #	x Supply (m)	y Supply (m)	x Return (m)	y Return (m)	ID #	x Supply (m)	y Supply (m)	x Return (m)	y Return (m)
MC0	2.025	1.775	2.025	1.775	MC18	3.625	0.425	2.02	1.775
MC1	2.025	1.775	2.025	0.425	MC19	3.625	0.425	2.02	0.425
MC2	2.025	1.775	3.625	1.775	MC20	3.625	0.425	3.625	1.775
MC3	2.025	1.775	3.625	0.425	MC21	3.625	0.425	3.625	0.425
MC4	2.025	1.775	0.425	1.775	MC22	3.625	0.425	0.425	1.775
MC5	2.025	1.775	0.425	0.425	MC23	3.625	0.425	0.425	0.425
MC6	2.025	0.425	2.025	1.775	MC24	0.425	1.775	2.025	1.775
MC7	2.025	0.425	2.025	0.425	MC25	0.425	1.775	2.025	0.425
MC8	2.025	0.425	3.625	1.775	MC26	0.425	1.775	3.625	1.775
MC9	2.025	0.425	3.625	0.425	MC27	0.425	1.775	3.625	0.425
MC10	2.025	0.425	0.425	1.775	MC28	0.425	1.775	0.425	1.775
MC11	2.025	0.425	0.425	0.425	MC29	0.425	1.775	0.425	0.425
MC12	3.625	1.775	2.025	1.775	MC30	0.425	0.425	2.025	1.775
MC13	3.625	1.775	2.025	0.425	MC31	0.425	0.425	2.025	0.425
MC14	3.625	1.775	3.625	1.775	MC32	0.425	0.425	3.625	1.775
MC15	3.625	1.775	3.625	0.425	MC33	0.425	0.425	3.625	0.425
MC16	3.625	1.775	0.425	1.775	MC34	0.425	0.425	0.425	1.775
MC17	3.625	1.775	0.425	0.425	MC35	0.425	0.425	0.425	0.425

Table 3. ID and locations of the occupant locations.

ID	Location (m)	ID	Location (m)
OL0	(1.25, 1.7, 1)	OL5	(3.75, 1.7, 2)
OL1	(2.5, 1.7, 1)	OL6	(1.25, 1.7, 3)
OL2	(3.75, 1.7, 1)	OL7	(2.5, 1.7, 3)
OL3	(1.25, 1.7, 2)	OL8	(3.75, 1.7, 3)
OL4	(2.5, 1.7, 2)		

The turbulent flow has fine velocity fluctuations which require fine space resolution or specific turbulence models for numerical simulations. To model turbulent flows, two common treatments are used: introducing turbulent model equations (Reynolds Averaged Navier-Stokes Modeling (RANS)), or using small-scale space-filter (large-eddy simulations) (Calautit, Hughes, and Nasir 2017; Hussain and Oosthuizen 2012; Kato 2018; Perén et al. 2015). RANS is a popular treatment method for CFD simulation of buildings. Here, a k-ε Reynolds averaged Navier-Stokes modeling (RANS) turbulent model is used to capture the turbulence effects within the model (Calautit, Hughes, and Nasir 2017; Hussain and Oosthuizen 2012). The Buoyancy effects are considered only in the gravitational term of the equations, i.e. the Boussinesq approximation is used.

We use OpenFOAM to numerically solve the nonlinear equations and find the velocity, pressure, and temperature in the room through time. OpenFOAM is an open-source solver which uses finite volume discretization. The PIMPLE algorithm which is a predictor-corrector iterative technique and combines semi-implicit methods for pressure linked equations (SIMPLE) and Pressure Implicit Split Operator (PISO) algorithms is employed to obtain the solutions.

Simulation and analysis process

The transient turbulent CFD analysis based on finite volume discretization with an element size of 0.05 m was conducted. The time step was defined variable to ensure Courant number is lower than 0.5 in each time step. The transient analysis simulated the simplified-building for a duration of 5 minutes. The cool air with a velocity of 0.75 m/s was supplied into the room during the simulation period. The supply air velocity ensures that the inside air can be exchanged up to 20 times per hour; the 5-minute simulation period is selected to make sure the inside air could be exchanged at least once during simulation. The cool air and the initial inside air temperatures respectively were 17 ºC and 27 ºC; the window was simulated with a constant temperature of 32 ºC. This study intended to investigate a case with extreme temperature differences, within the acceptable range of Boussinesq approximation (15 ºC for air) (Ferziger and Perić 2012), which might not reflect realistic conditions of commercial buildings. Yet ASHRAE Fundamentals Handbook section (20.10) suggests a maximum temperature difference of about 8 ºC between supply and the room for cooling when the air directs horizontally and from a location near the ceiling (ASHRAE 2009), which is not far from the assumption made. We acknowledge the importance of relative humidity for indoor air quality, however, this study only focuses on temperature and velocity variations based on the mechanical configuration and occupant location.

To perform the analysis, a parametric model, called *template*, was made in OpenFOAM and employed for the automation of the simulation process (Weller et al. 1998). Python scripts were developed to set up all simulation cases from the template folder, automate the geometry and mesh generations processes, and initiate the solving procedure of OpenFOAM. In the simulation process, the results in pre-defined locations for all time steps were collected and reported in files which were used for further post-processing done in Python. ParaView is used for the visualization of 3-dimensional temperature and velocity fields. (Ahrens, Geveci, Law, Charles 2005).

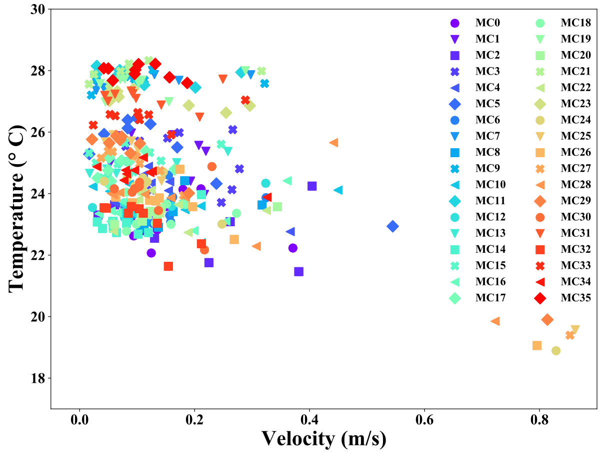
RESULTS

Results are presented in two sections. Section 1 mainly focuses on investigating the temperature and velocity variations, and Section 2 analyzes the relative placements of supply-return.

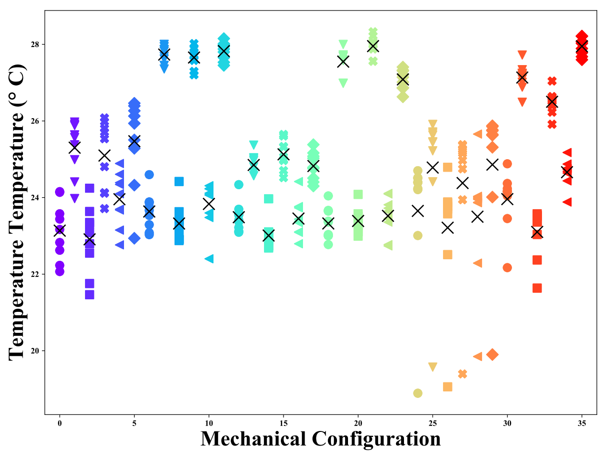
Temperature and air velocity predictions

Figure 2 presents the temperature and the velocity values obtained for all 324 combined configurations (6 supply × 6 return × 9 occupant locations). The values obtained for each mechanical configuration, i.e. 9 occupant locations, are plotted with the same color-symbol combination. Figure 2a displays the temperature vs. velocity values; the results indicates that the velocity and temperature data are mainly clustered in the left-side of the figure. Yet, a number of combined configurations with low temperature values have high velocity (>0.6m/s). For these cases, the cool air with high velocity flows to the occupant locations which is uncomfortable.

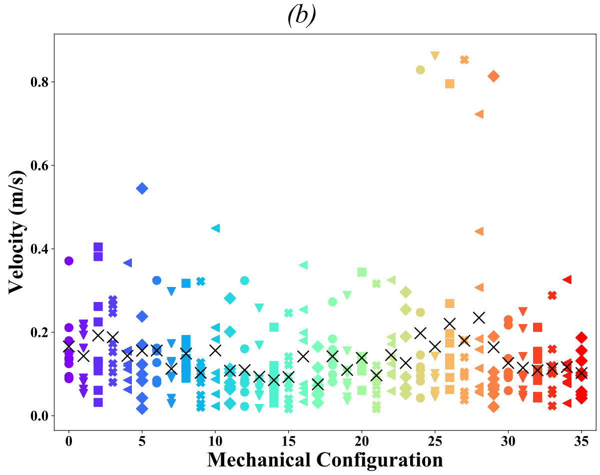
Figure 2b and 2c respectively illustrate the temperature and velocity values obtained for all combined configurations. The values corresponding to each mechanical configuration are plotted vertically. The average values (over all 9 occupant locations) are also presented in these plots with black cross-markers for comparison purpose.



(a)



(b)



(c)

Figure 2 (a) The temperature vs. velocity for all 326 combined configurations. The (b) temperature and (c) velocity for all mechanical configurations. The values corresponding to each mechanical configuration are plotted with the same color-symbol combination.

Figure 3 presents the same information as Figure 2 in an aggregated form to ease interpretation. Figure 3a. shows the average temperature vs. the average velocity for each of the 36 mechanical configurations. The violins plots in Figures 3b and 3c show the average values, extrema, and probability densities for temperature and velocity respectively. Comparing all mechanical configurations, the average temperatures and velocities vary about 5.04 ºC and 0.16 m/s.

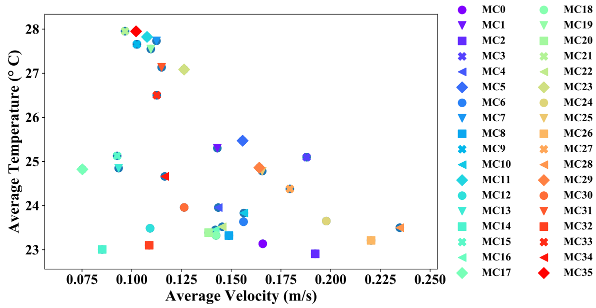
As shown in Figure 3b, the average temperatures of 17 mechanical configurations are lower than 24 ºC, the arbitrary maximum temperature threshold we use for interpretation purposes only. This result shows the impact of the mechanical configuration on the average temperature while the input energy level is fixed. Additionally, the extrema and the probability density illustrate, while an average temperature lower than 24 ºC can be achieved, the temperature of several occupant locations could be higher than 24 ºC, see MC4, MC10, and MC28. The extrema shown in the figure illustrate that the temperature differences in different mechanical configurations can vary considerably, for example 6.34 ºC obtained for configuration MC25.

In Figure 3c, the horizontal line plots the velocity at an arbitrary 0.25 m/s threshold. The average velocities of all mechanical configurations are lower than 0.25 m/s. Yet for several cases the velocities at different occupant locations are higher than 0.25 m/s and for 6 combined configurations the velocity is higher than 0.6 m/s (refer to the probability density of the data and Figure 2c). The maximum and minimum velocity differences are 0.817 m/s and 0.134 m/s for configuration MC25 and MC17.

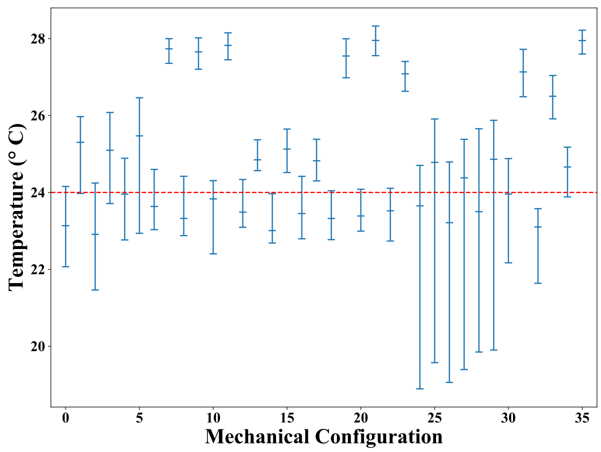
Analysis of configurations

Five different configuration categories, classified based on their temperature and acceptance ratio, are discussed in the following. The acceptance ratio, a measure defined for configuration classification, is the percentage of the occupant locations with the temperature and velocity lower than 24 ºC and 0.25 m/s, respectively. Table 4 presents a number of these configurations and their corresponding data.

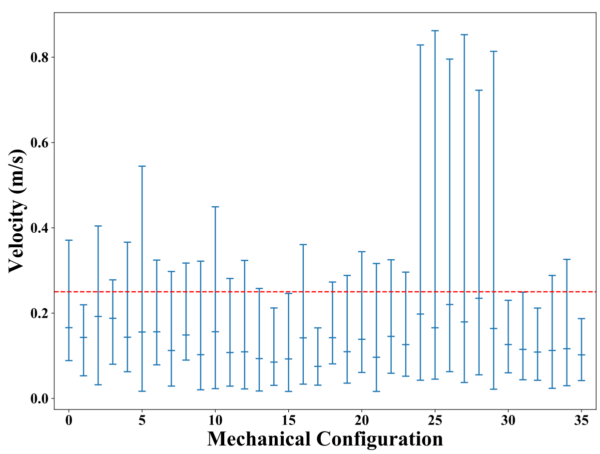
Minimum average temperature. Configuration MC2 has the lowest average temperature which is consistent with the ASHRAE discussion on the outlet (supply) (called outlet classification, page 20.7—9). However, its temperature difference (2.78 ºC) is higher than 28 other configurations; the average temperatures of 13 of these cases are lower than 24 ºC. For MC2, the supply is placed near the ceiling and in the middle section of the building and the return is located near the wall oposite the window and ceiling.



(a)



(b)



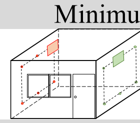
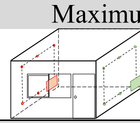
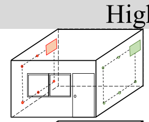
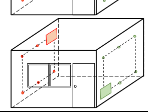
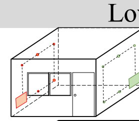
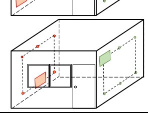
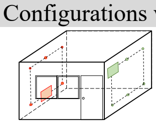
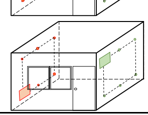
(c)

Figure 3 (a) The variation of average temperature vs. average velocity of the mechanical configurations. The average, extrema, and probability density of (b) temperature and (c) velocity.

High temperature and velocity acceptance ratios (≥ 89%). For all the cases, the return is placed near the ceiling yet the supplies are located near the floor or the ceiling. Both the return and supply are not located near the window (MC6, MC8, MC12, MC18, and MC20).

High average temperature (≥ 26℃). For all these cases, the temperature acceptance ratio is 0% and both the supply and return are located near the floor in different horizontal locations, which means cool air enters and exits the room while moving near the floor without being very well mixed with the warm air (MC7, MC9, MC19, MC21, MC23, MC33, and MC35). The results are consistent with ASHRAE suggestions that a supply located near the floor which direct the flow horizontally into the room are not recommended for summer cooling (page 20.11). A large stagnant zone could be formed in the entire upper region of the room.

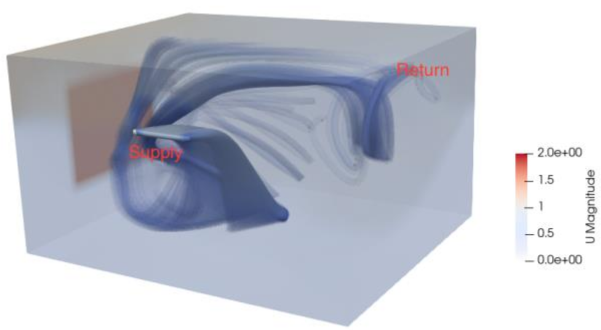
Table 4 Information on the mechanical of configurations with the minimum and maximum average temperatures, low and high thermal acceptance ratio, and high temperature variation. In the table, Conf = configuration, Ave=average, Accep=Acceptance ratio.

Conf	Schem	Ave T	Temp STDE	Ave U	Accep T %	Accep U %
Minimum average temperature						
2		22.9	0.8	0.19	89	67
Maximum average temperature						
21		28.0	0.2	0.10	0	89
High acceptance ratio						
14		23.0	0.4	0.085	100	100
32		23.1	0.6	0.109	100	100
Low acceptance ratio						
23		27.1	0.2	0.126	0	78
9		27.7	0.2	0.103	0	88
Configurations with high temperature variation						
24		23.7	1.7	0.199	22	89
29		24.9	1.8	0.165	11	89

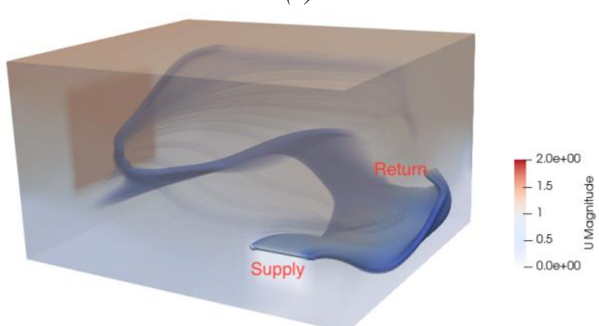
High temperature variation (around 6℃*).* For all the cases, the supply is placed near the window and the ceiling while the return can be located in all 6 defined positions for the return (MC24 to MC29). This finding is not in agreement with the ASHRAE suggestion to direct cool air toward the heat source (page 20.10).

Summarizing observations on the various categories of configurations, the relative locations and distances of supply with respect to the window and return, as well as supply and return heights, play important roles in temperature and velocity variations and their averaged values.

Figure 4 illustrates 3-dimensional representations of configurations MC2 and MC21, respectively, with the minimum and maximum average temperatures. The background room color shows the air temperature and the streamlines show the airflow patterns and speed. The streamlines shown in the picture start from a horizontal line in the inlets. Their purpose is to show the airflow patterns and mixtures in the room.



(a)



(b)

Figure 4 (a) Configuration MC2 with the minimum average temperature. (b) Configuration MC21 with the maximum average temperature. Background room color shows the air temperature, where red represents warmer areas, and the streamlines show the airflow direction and speed.

DISCUSSION

This research considers the possibility of increasing energy efficiency and enhancing occupant comfort through architectural, interior, and mechanical system design decisions collectively informed by high-fidelity simulations. The results obtained show how mechanical configurations affect the temperature and velocity variations through an indoor space. The results indicate the same forced-air cooling input may produce location-specific temperatures and velocity that differ by as much as 9.4 ºC and 0.71 m/s depending on the overall configuration. Additionally, the average temperature and velocity may vary about 5.0 ºC and 0.16 m/s depending on the mechanical configuration.

As stated at the outset, most building energy modeling tools are based on the well-mixed zone air assumption. This means the exact placement of building elements such as supply and return vents, windows, room partitions, and furniture, have little impact on energy efficiency and occupant comfort, and are therefore simplified or neglected in conventional models. The results of this study show that average values are not always good representatives of the thermal and velocity characteristics of a space. Depending on the building, it is possible that a reliance on the well-mixed assumption may obscure opportunities to improve the configuration building elements in a way that appreciably improves energy efficiency or comfort.

Further investigation of element placements for different configurations reveal interesting points. The results for MC14 is surprising because the supply and the return of this configuration are directly facing each other, which means this design might not be the primary configuration that a designer considers. Additionally, for all cases with high acceptance ratios ($\geq$ 89%), the return is placed near the ceiling yet the supplies are located near the floor or the ceiling; as mentioned above, while the return location is in agreement with ASHRAE guidelines, the supply location does not follow the recommendations about placing the supply near the floor. For these cases, the return and supply are not located near the heat source, i.e. the window. Additionally, for all cases with high temperature variations (around 6), the supply is placed near the window and the ceiling, which is not consistent with ASHRAE's recommendations about supply placement as discussed before; the return is located in all 6 pre-defined positions. While the results for a number of cases are in agreement with ASHRAE's guidelines on supply and return locations, the findings suggest that temperature and velocity variations are complex functions of relative element placement and need deeper examination. It should be noted that the performance analysis provided by ASHRAE (section 20) is mainly based on airflow patterns (ASHRAE 2009).

This study is a step toward exploring the possible range of effects the architectural, occupant, and combined configurations can have on energy efficiency and occupant comfort. Further investigation on the temporal data at occupant locations could reveal new aspects of design and comfort. Moreover, an important aspect for building design is their resilience and adaptiveness to changes as occupants continuously interact with buildings. The system energy consumption and life-cycle costs are functions of these interactions and changes. Optimizing and controlling the temperature and the velocity locally, where occupants spend most of their time, might result in lower energy use. High-fidelity CFD analyses could provide means to better manage and control buildings under these changes. The direction of the airflow, air temperature, and air speed are among other CFD parameters which could be controlled and adjusted in a smart building based on occupant locations and their interactions with the building. To develop a full picture of the problem, additional studies will be needed to establish the viability of combining HVAC and CFD analyses for improving occupant comfort and reducing energy consumption of build environments; the topic has been receiving attention (Berquist et al. 2017; Kato 2018; Zhai and Chen 2005).

This work only offers limited aspects of thermal comfort analysis. For the sake of simplicity, the placement of thermostats and the building automation system have been excluded. Whereas the study showed large discrepancies in temperature, a more comprehensive model might instead predict large discrepancies in energy consumption as the system strives to meet a setpoint. This study also uses simplified assumptions for the boundary conditions and numerical modeling. For example, only one air velocity was used at the supply. The effect of assumptions and parameters of the CFD simulation and how to improve the simulation accuracy for particular applications need to be further investigated. This research has tried to isolate the effects of temperature and velocity variations due to element placements, throughout the space. The comparisons of the results with indices aggregating different data types, for example Dry Radiant Temperature, may reveal interesting information for designers (Levermore 2000). These indices summarize information and could be practical for designers. Finally, a limited number of architectural configurations have been examined. Exploring the effect of radiation and heat conduction through the walls using conjugate heat transfer models will also provide more information and could clarify more aspects of this complex problem. The aim of this investigation is to provide insight about the potential effect of combined configurations on the energy and comfort levels.

CONCLUSION

This research examines to what extent designers and engineers could save energy and improve comfort by employing high-fidelity CFD simulations when planning the arrangement of building elements. By simulating 36 configurations of forced-air cooling supply and return vents, and extracting predictions at 9 possible occupant locations, the study shows how element placement can affect the temperatures and air velocities experienced by building occupants. Although most building energy modeling tools are based on the well-mixed zone air assumption, under which the exact placement of combined configuration elements have little impact on energy efficiency and occupant comfort, the results obtained clarify that the mechanical configuration is significant, and that the temperature and velocity of one occupant location is not necessarily representative of a space's thermal characteristics. In particular, the results show that the relative locations and distances of supply with respect to the window and return as well as supply and return heights play important roles in temperature and velocity variations and their averaged values.

Further sensitivity studies, which take the effects of the placement and size of elements into account, could provide more insight for designing highly energy efficient and comfortable buildings. Also, future work is needed to determine the technological requirements and viability of incorporating CFD and other high-fidelity analyses into earlier stages of the design process, so that all aspects of a design can be simultaneously optimized with knowledge of the resulting air flow patterns and their implications.

REFERENCES

Ahmed, AQ, and Shian G. 2017. "Numerical Investigation of Height Impact of Local Exhaust Combined with an Office Work Station on Energy Saving and Indoor Environment." *Building and Environment* 122: 194–205.

Al-Sanea, SA, MF Zedan, and MB Al-Harbi. 2012. "Effect of Supply Reynolds Number and Room Aspect Ratio on Flow and Ceiling Heat-Transfer Coefficient for Mixing Ventilation." *International Journal of Thermal Sciences* 54: 176–87.

Ahrens, J, Geveci, B, Law, C, "*ParaView: An End-User Tool for Large Data Visualization*, Visualization Handbook", Elsevier, 2005

ASHRAE. 2009. "ASHRAE Handbook of Fundamentals." New York. The American Society of Heating, Refrigerating and Air-Conditioning Engineers.

Berquist, J, Tessier, A, O'Brien, W, Attar, R, and Khan, A. 2017. “An Investigation of Generative Design for Heating, Ventilation, and Air-Conditioning.” *Proceedings of the Symposium on Simulation for Architecture and Urban Design. Society for Computer Simulation International.*

Calautit, JK, Hughes, BR, and Nasir DS. 2017. “Climatic Analysis of a Passive Cooling Technology for the Built Environment in Hot Countries.” *Applied Energy* 186: 321–35.

Ferziger, JH, and Perić, M. 2012. *Computational Methods for Fluid Dynamics*. Berlin: Springer International Publishing.

Gilani, S., Montazeri H., and Blocken B. 2016. “CFD Simulation of Stratified Indoor Environment in Displacement Ventilation: Validation and Sensitivity Analysis.” *Building and Environment* 95: 299–313.

Al Horr, Y, Arif, M, Kaushik, A, Mazroei, A, Katafygiotou, M, and Elsarrag, E. 2016. “Occupant Productivity and Office Indoor Environment Quality: A Review of the Literature.” *Building and environment,* 105: 369–89.

Al Horr, Y, Arif, M, Kaushik, A, Mazroei, A, Elsarrag, E, and Mishra, S. 2017. “Occupant Productivity and Indoor Environment Quality: A Case of GSAS.” *International Journal of Sustainable Built Environment* 6(2): 476–90.

Gorissen, D, Dhaene, T, and De Turck, F, 2009. “Evolutionary model type selection for global surrogate modeling.” *Journal of Machine Learning Research*, 10(71): 2039-2078.

Hussain, S, and Oosthuizen PH. 2012. “Numerical Study of Buoyancy-Driven Natural Ventilation in a Simple Three-Storey Atrium Building.” *International Journal of Sustainable Built Environment* 1(2): 141–57.

Kanaan, M. 2019. “CFD Optimization of Return Air Ratio and Use of Upper Room UVGI in Combined HVAC and Heat Recovery System.” *Case Studies in Thermal Engineering* 15: 100535.

Kato, Shinsuke. 2018. “Review of Airflow and Transport Analysis in Building Using CFD and Network Model.” *Japan Architectural Review* 1(3): 299–309.

Kim, D, Braun, JE, Cliff, EM, and Borggaard, JT. 2015. “Development, Validation and Application of a Coupled Reduced-Order CFD Model for Building Control Applications.” *Building and Environment* 93: 97–111.

Lee, JH. 2007. “Optimization of Indoor Climate Conditioning with Passive and Active Methods Using GA and CFD.” *Building and environment* 42(9): 3333–40.

Levermore, GJ. 2000. “Building energy management systems: an application to heating and control.” Taylor & Francis.

Mahdavi, A. 2020. “In the Matter of Simulation and Buildings: Some Critical Reflections.” *Journal of Building Performance Simulation* 13(1): 26–33.

Mendell, MJ, Fisk, WJ, Kreiss, K, Levin, H, Alexander, D, Cain, WS, Girman, JR, Hines, CJ, Jensen, PA, Milton, DK and Rexroat, LP. 2002. “Improving the Health of Workers in Indoor Environments: Priority Research Needs for a National Occupational Research Agenda.” *American journal of public health* 92(9): 1430–40.

Moosavi, L, Mahyuddin, N, Ab Ghafar, N, and Ismail, MA. 2014. “Thermal Performance of Atria: An Overview of Natural Ventilation Effective Designs.” *Renewable and Sustainable Energy Reviews* 34: 654–70.

Perén, JI, van Hooff, T, Leite BC, and Blocken B. 2015. “CFD Analysis of Cross-Ventilation of a Generic Isolated Building with Asymmetric Opening Positions: Impact of Roof Angle and Opening Location.” *Building and Environment* 85(0): 263–76.

Sakai, K, Kubo, R, Kajiya, R, Iwamoto, S, Kurabuchi, T, and Kishida, T. 2008. “CFD Analysis of Thermal Environment of a Room with Floor Heating or Air Conditioning.” *Indoor Air 2008, 17-22 August 2008, Copenhagen, Denmark - Paper ID: 315 CFD* (August): 17–22.

Sinha R, Lennartsson M, Frostell B. 2016. “Environmental Footprint Assessment of Building Structures: A Comparative Study.” Building and Environment 104: 162–71.

Stavridou, AD, and Prinos PE. 2017. “Unsteady CFD Simulation Ina Natural Ventilated Room with a Localized Heat Source.” *Procedia environmental sciences* 38: 322–30.

Weller, HG, Tabor G, Jasak H, and Fureby C. 1998. “A Tensorial Approach to Computational Continuum Mechanics Using Object-Oriented Techniques.” *Computers in physics* 12(6): 620–31.

Yang, X, Zhong, K, Kang, Y, and Tao, T. 2015. “Numerical Investigation on the Airflow Characteristics and Thermal Comfort in Buoyancy-Driven Natural Ventilation Rooms.” Energy and Buildings 109: 255–66.

Zhai, Z. 2006. “Application of Computational Fluid Dynamics in Building Design: Aspects and Trends.” *Indoor and Built Environment* 15(4): 305–13.

Zhai, ZJ, and Chen, QY. 2005. “Performance of Coupled Building Energy and CFD Simulations.” Energy and Buildings 37(4): 333–4