Using trickle ventilators coupled to fan extractors to achieve a suitable airflow rate in an Australian apartment: a CFD modelling approach.

Mikael Boulic[1*], Pierre Bombardier[2], Zain Zaidi[3], Andrew Russell[4] and David Waters[5]

1 School of Built Environment, Massey University Auckland Campus, Auckland, New Zealand
**Corresponding author: m.boulic@massey.ac.nz*

2 Faure QEI, ATRIX Group, France

3 Dynamic Composite Technologies Pty Ltd, Australia

4 Proctor Group Australia, Australia

5 APL Window Solutions, New Zealand

ABSTRACT

The level of airtightness is increasing in newly built Australian apartments. An appropriate ventilation rate is needed to provide occupants with a healthy environment. In 2022, a significant proposed change in the Australian National Construction Code (NCC) would require building tested as achieving less than five air changes per hour at 50Pa to have a continuous flow exhaust. As occupants tend to not open windows, there is a need to inform about the potential benefit of using standard or innovative trickle ventilators (opening variation with real-time conditions) to assist occupants in ventilating their apartment. Due to the COVID pandemic, restrictions have forced many people to work from home. Under these circumstances, it is crucial to ensure that occupants are exposed to a healthy environment when spending more time at home. The airflow in a Melbourne apartment occupied by two adults and a child over three hours was modelled using a computational fluid dynamic approach and following the proposed NCC requirements. The results showed some local effects, which depend on temperature gradients, and source locations in the airflow.

KEYWORDS

Ventilation, carbon dioxide, occupied apartment, Computational Fluid Dynamic (CFD), IAQ, trickle ventilator

1 INTRODUCTION

The Australian National Construction Code (NCC) require residential buildings to have thermal resistance and ventilation of spaces where moisture could be generated. Still, neither recommends achieving any minimum moisture level. Under health and amenity (Volume 2 Part 3.8.5), the NCC requires that "the window, opening, door or other device have a ventilating area of not less than 5% of the floor area". The NCC relies on occupants to open windows to reach a suitable ventilation level. This dependence on occupants for natural ventilation could only be acceptable in houses built before 2000. Basset (2001) reported that pre-2000 residential New Zealand buildings showed high values from infiltration rate measurements (equivalent to a low airtightness for these pre-2000-built buildings). However, McNeil et al. (2015) found that residential buildings constructed after 2000 were much more airtight (change in building materials and construction methods). Lower infiltration rates were found for more recently built buildings, making additional ventilation (natural or mechanical) necessary to dilute moisture and potential pollutants. The same authors reported a measured ventilation rate in a third of post-2000 houses very close to the estimated infiltration rates from blower door infiltration tests, which indicated that the occupants were not opening windows that often (McNeil et al., 2015). These findings challenge occupants' ability to open windows to reach a suitable ventilation level in post-2000 homes. NCC states that a mechanical ventilation system will be required where 5% of the floor area of openable windows cannot be met. Mechanical ventilation is seldom installed in these recently built houses, and occupants are not opening windows as the NCC requires. Australian and New Zealand buildings are constructed using similar construction methods and materials, so we expect Australian houses built after 2000 to be more airtight and seldomly open windows as in New Zealand. Due to the COVID pandemic, restrictions have forced many people to work from home. Under these circumstances, it is crucial to ensure that occupants are exposed to a well-ventilated environment when spending more time at home.

In 2022, a significant proposed change in NCC would require homes, tested to achieve less than five air changes per hour at 50 Pa, to provide a mechanical ventilation system with a flow rate (Q) of not less than achieved following Equation 1.

$$Q = (0.05 \times A + 3.5 \times (N + 1)) / p \tag{1}$$

Where:
Q = the required air flow rate (L/s)
A = the total area of the building (m^2)
N = the number of bedrooms in the building
p = the fraction of time within each four-hour segment that the system is operational

Using a Computational Fluid Dynamic (CFD) approach, the airflow in a Melbourne apartment, occupied by two adults and a child for over three hours, was modelled following these proposed NCC requirements. Carbon dioxide (CO_2) was used as a tracer gas to determine the ventilation parameters but not as an overall indicator of the indoor air quality in the apartment (Persily, 2020).

This paper introduces a CFD modelling approach to investigate suitable ventilation solutions within the proposed NCC 2022 requirements.

2 MATERIALS AND METHODS

2.1 Description of the building and ventilation

Modelled apartment building
A two-bedroom apartment, located in Melbourne (Australia), was used in this simulation. Figure 1 (left) shows the 3D model imported to the CFD program. This 90 m^2 floor apartment is located on the 21^{st} floor of a 45-storey building. The primary bedroom, living room, office and child bedroom have all north-facing windows (potential solar gain due to large glazing surfaces).

Ventilation
The fan extractors are in the primary bathroom (continuous flow rate of 10 L/s), the kitchen (continuous flow rate of 12 L/s) and the child bathroom (continuous flow rate of 10 L/s). This flow rate (total 32 L/s) follows the proposed NCC 2022 minimum requirements (Equation 1). Five trickle ventilators are installed on the north-facing windows (one in the primary bedroom,

two in the living area, one in the office, and one in the child's bedroom). All five trickle ventilators have the same size of 700 mm by 55 mm (108.0 cm^2 open area or 32.2 cm^2 effective open area). The air will enter the apartment via the five trickle ventilators and be extracted via the three fan extractors (cross-ventilation) as displayed in Figure 1, right-hand side.

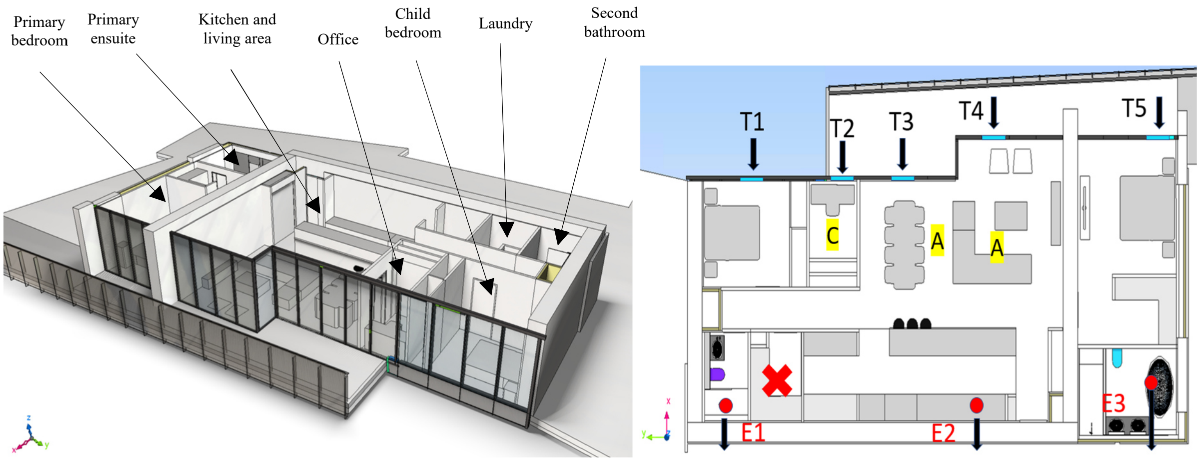

Figure 1: 3D CAD model of the two-bedroom apartment (left) and floor plan with cross-ventilation features, E1-E3: Three fan extractors (south), T1-T5: five trickle ventilators on north-facing windows. The laundry area has a red cross (X) because the laundry volume was removed to simplify our first CFD simulations (right). The child (C) is in the office, and the two adults (A) are in the living room.

Occupants, activities, and carbon dioxide generation rate

In this modelling, the two-bedroom apartment is occupied by two adults (male, 40 years old and female, 40 years old) and one child (10 years old). They are undertaking light effort tasks (MET 1.5) for three hours. The two adults (A) are in the living area, and the child (C) is in the office area, as shown in Figure 1, right-hand side. Table 1 shows the CO_2 generation rate (L/s) for each of the three occupants during sitting tasks/light activity (MET 1.5).

Table 1: Carbon dioxide generation rate (L/s) at MET 1.5: sitting tasks, light effort (Persily et al., 2017)

Occupants	Source location	Generation rate (L/s) at MET1.5
Male (40 years old)	Living room	0.0058
Female (40 years old)	Living room	0.0045
Child (10 years old)	Office	0.0037

The CO_2 sources are defined by well-mixed volumes (Figure 2), where the gas is generated at a constant rate. These yellow volumes in Figure 2 are the locations with a good likeliness of presence (assuming where people are likely to stand or sit in the living room and the office).

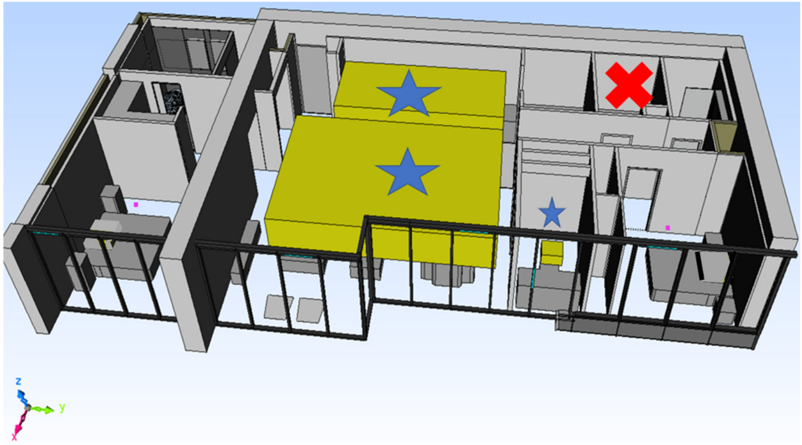

Figure 2: Location of the CO_2 sources (blue stars) in the apartment (two adults in the living area and one child in the office area).

2.2 CFD settings and parameters

The general equations used in this CFD simulation can be found in Versteeg et al. (2007). These equations are based on the mass, energy, and momentum conservation laws. The CFD modelling was undertaken using a commercial CFD software, scSTREAM (Cradle CFD part of Hexagon Manufacturing Intelligence, Hexagon AB Group, Sweden).

Turbulence models

CFD employs turbulence models to simulate fluid motion and the appearance of eddies. These turbulence models are specific to different situations, and a significant research effort has been dedicated to their development (Stevanovic, 2009). In ventilation studies, air turbulence is usually described using the Reynolds-Averaged-Navier-Stokes (RANS) equations (Yakhot, 1986; Lu et al., 2019). Derived from the RANS equations, several models have been developed, like the ReNormalization Group (RNG) K-epsilon turbulence model and Linear Low Reynolds (LLR) model. They are commonly used for airflow simulation in an indoor environment (Hassan, 2017). The LLR model (Abe et al., 1994) improves the prediction accuracy in the near-wall region, an essential aspect of ventilation at a close range of surfaces. This study used the LLR model to simulate indoor airflows expecting low-velocity airflows generated near surfaces (trickle ventilators are often located near the ceiling).

Heat sources and heat transfer

The preliminary simulation did not consider solar gain from the large north-facing glazing surfaces (Figure 1) and heat gain from occupants. However, this will be considered in future work.

Boundary conditions

Air inlet

Figure 3 shows the profile of the trickle ventilator model (APL Ventient SCW-SH700) used in the CFD simulation. This model is representative of the horizontal-flow-type of trickle ventilator and includes a shape memory alloy adapting the opening to external temperature (fully open above 18°C and minimum opening under 12°C). Section 2.1 - Ventilation mentioned that five trickle ventilators were installed on the north-facing windows. A 700 mm trickle ventilator was adapted to the window frame dimensions. It incorporates a passive wind dampener to manage water ingress and draughts associated with high wind gusts. The 500 mm to 1500 mm range models and vertically oriented trickle ventilators will be tested in the future.

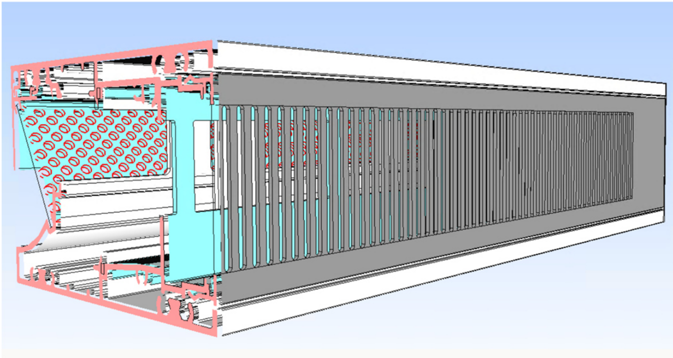

Figure 3: 3D model of one APL Ventient SCW-SH700 (Trickle ventilator)

Air outlet
In the first simulations, the extracted airflow is imposed to the regulatory value (Section2.1). Still, it will be controlled by the fan Q (air volume) /P (static pressure) curve in the subsequent development.

Wind force
Under a continuous extraction regime (use of the three fan extractors), the air flowing through the trickle ventilators will be influenced by the dynamic pressure applied due to the wind force. This influence was assessed with a large-scale wind simulation (Figure 6). The wind velocity profile obtained on the building windward surfaces was tested. A 4.5 m/s (16 km/h) wind velocity was used, corresponding to the monthly average wind speed monitored at Melbourne Essendon Field Airport for the 1995 - 2021 period (Iowa State University, 2022).

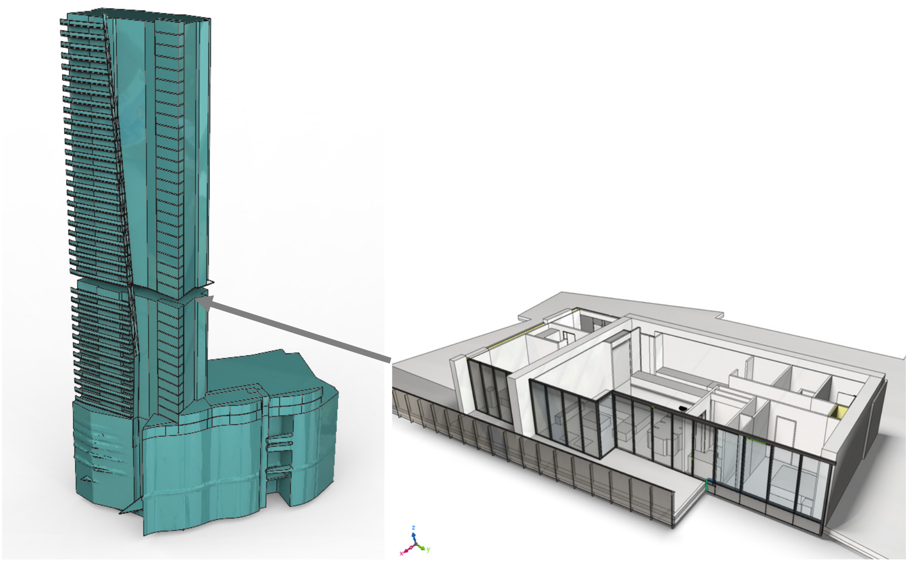

Figure 4: Building with the location of the apartment (21st floor of the 45-storey building)

Walls
A logarithmic scale is applied to the surfaces to consider the air friction stress (turbulent conditions).

Computational grid and convergence criteria
scSTREAM has a finite volume discretisation scheme. The computational grid is a structured hexahedral mesh defined following CFD good practices (Sørensen et al. 2003). A sensitivity analysis was run after refining the volumes where considerable pressure or velocity gradients were expected, and then the grid convergence was obtained. The solver convergence was

assessed by meeting residual error convergence criteria (10^{-6} level) and stability for monitoring points (pressure velocity and CO_2). The final mesh includes 6 to 8 million nodes for the models described below.

2.3 Simulation Methodology

Figure 5 shows the cascade model followed during our 3-step simulation. The first simulation, called the "Wind Model", will define the velocity range on the 21st floor on the input face of the trickle ventilators for the main wind directions (North, South and South-West) following the Melbourne wind rose (Iowa State University, 2022). The results from this "Wind Model" will be implemented in the second simulation called the "Trickle vent Model", which will describe air diffusion accurately from the trickle ventilator depending on the in/out pressure and temperature difference. The velocity profile obtained will then be used in the third and last simulation. This final simulation stage will combine the results from the "Trickle vent Model" and the "Fan Q/P equation" to define the airflow behaviour in the apartment. This third model is called the "Apartment Model". The wind and trickle vent models were steady-state models, and the apartment model was simulated for three hours (transient model).

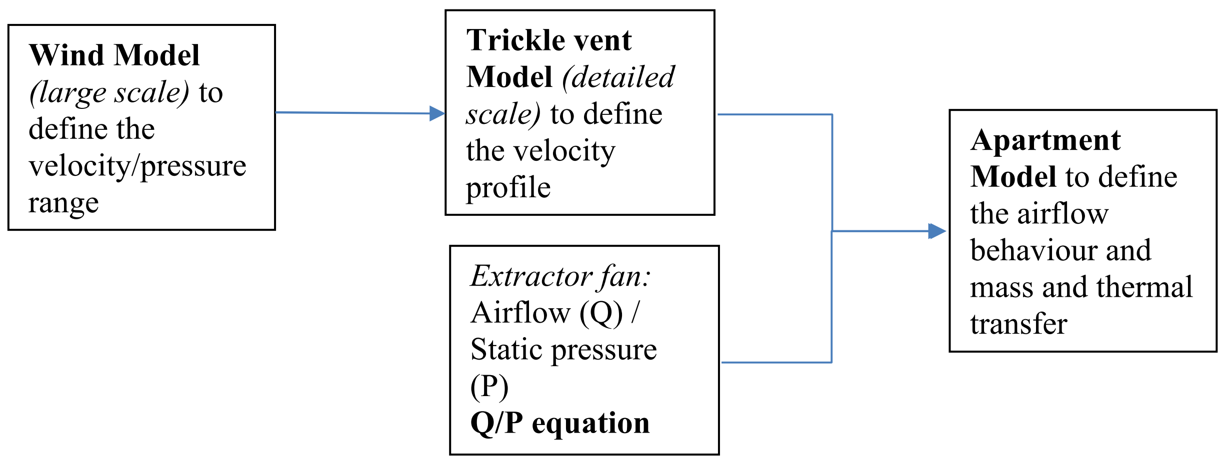

Figure 5: Conceptual framework of the CFD cascade model

2.4 Occupation scenario considered

The CO_2 generated by occupants inside the apartment during their light-level activities was of interest. The CFD simulation keeps the outside CO_2 background level constant (0 ppm). The results will be increased by 400 ppm (current outside CO_2 level at Melbourne). Unfavourable conditions were used in the first airflow model (maximum occupancy for this two-bedroom apartment estimated at three people, maximum period of occupancy estimated at three consecutive hours). The choices were directed by the size of the two-bedroom apartment (two adults and one child), and three hours could well reflect the current COVID pandemic context when people tend to stay longer at home.

3 RESULTS AND DISCUSSION

At this project stage, the three models are not yet connected. The following sections will present the three models separately (Section 3.1 Wind Model, Section 3.2 Trickle vent Model and Section 3.3 Apartment Model). In the future, the three models will be combined, as displayed in our conceptual framework.

3.1 Wind Model: defining the wind ranges on the building facade

Figure 6 shows a large-scale model (250 m width and 170 m height, eight million nodes) of the whole 45-storey building, informing of the velocity values. On the windward side of the building (the north-facing side in our simulation), the velocity ranges from 0 to 6.0 m/s, with an average value of around 2.4 m/s in the vicinity of our 21st floor selected apartment (located by the white strip in Figure 6).

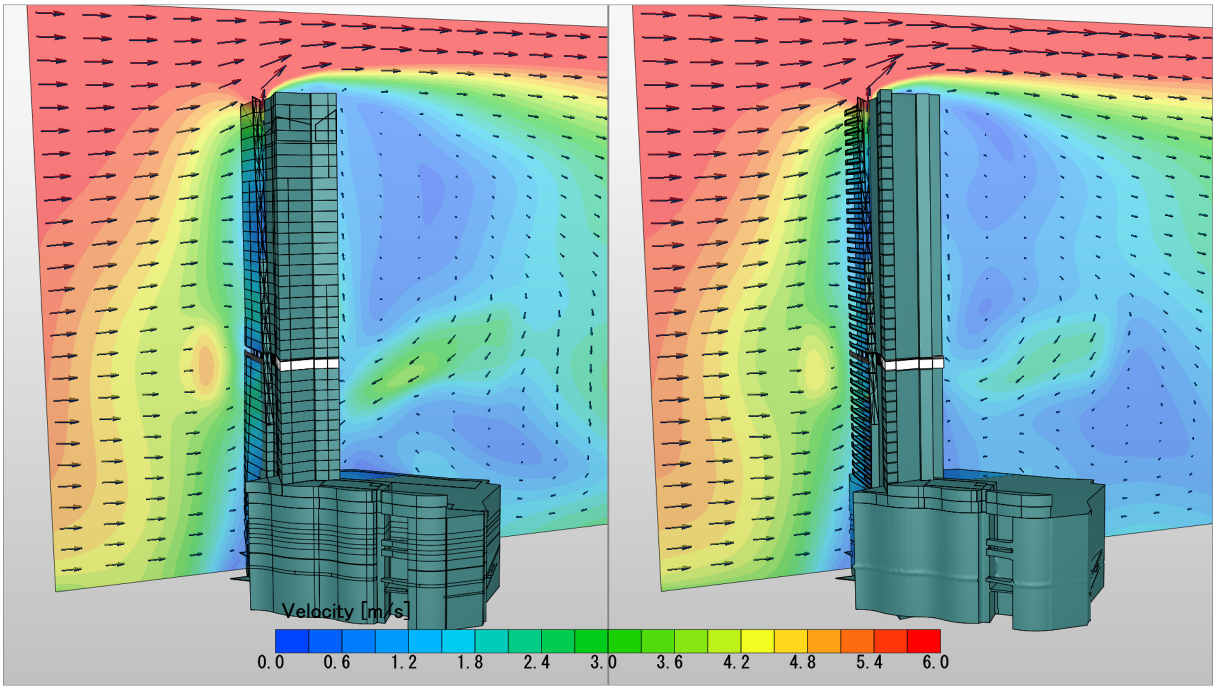

Figure 6: CFD modelling of the wind velocity on the façade of the 21st-floor balcony design only (left) and with the design of the 45-storey balconies considered (right).

The first simulation only considered the balcony design for the 21st studied floor (Figure 6, left). The second simulation considered all 45 balcony designs (Figure 6, right). The results showed minor disturbance in the vicinity of the 21st floor when all 45 balcony designs were considered.

3.2 Trickle vent Model: defining the air velocity profile through the ventilator.

It is essential to define the airflow profile through the trickle ventilator. This profile will be used as input in the "Apartment Model". Three temperature differences between intake and room temperature were tested in Case studies 1, 2 and 3.

- Case study 1: Spring or Autumn seasons (no difference between intake temperature and room temperatures - Figure 7),
- Case study 2: Winter season (intake temperature is 10°C cooler than the room temperature - Figure 8),
- Case study 3: Summer season (intake temperature is 10°C warmer than the room temperature -Figure 9).

Following the proposed NCC 2022 fan flow rate requirements, a total air intake of 32 L/s (10 L/s for the primary bathroom fan flow rate + 12 L/s for the kitchen fan flow rate + 10 L/s for the child bathroom fan flow rate) was obtained. As there are five trickle ventilators, each trickle ventilator will have a mean airflow rate of 6.4 L/s.

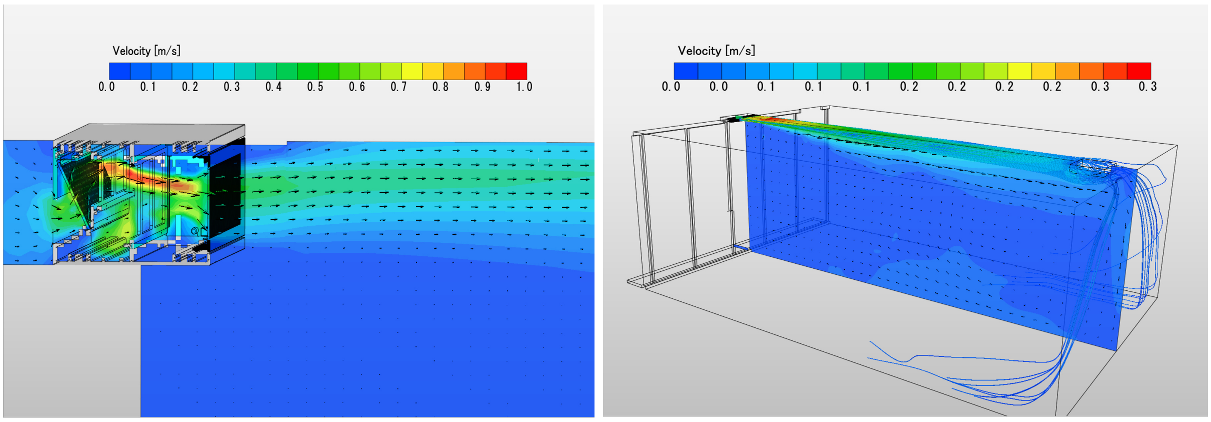

Figure 7: Case study 1 - Trickle ventilator CFD with air intake temperature equal to room temperature

Figure 7 shows that the streamlines of the airflows (coloured by velocity) exiting from the trickle ventilators stay close and parallel to the ceiling surface.

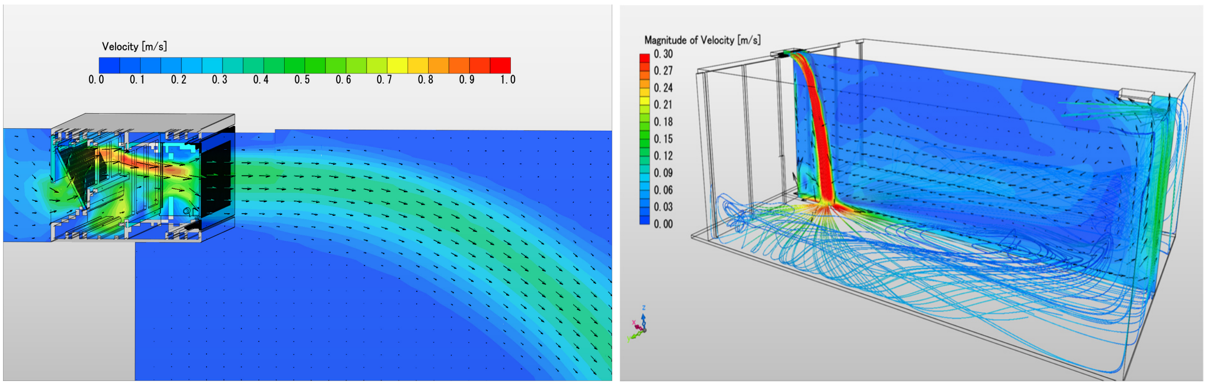

Figure 8: Case study 2 - Trickle ventilator CFD with air intake temperature 10°C cooler than room temperature

Figure 8 shows that the streamlines of the airflows exiting from the trickle ventilators quickly fall to the floor. Case study 2 will discomfort people sitting close to the windows where the trickle ventilator is installed. More work will be undertaken to investigate potential solutions to remove discomfort. For example, a heating source could be added to the system (convective heat transfer), and CFD modelling could inform the best heater source location. When air intake temperatures are cool, the tickle ventilator would partially close by passive operation of bimetallic shape memory alloy (SMA) springs to reduce discomfort. This was not accounted for in the study, but future modelling will examine this to help inform the correct set points for the SMA springs for optimising thermal comfort.

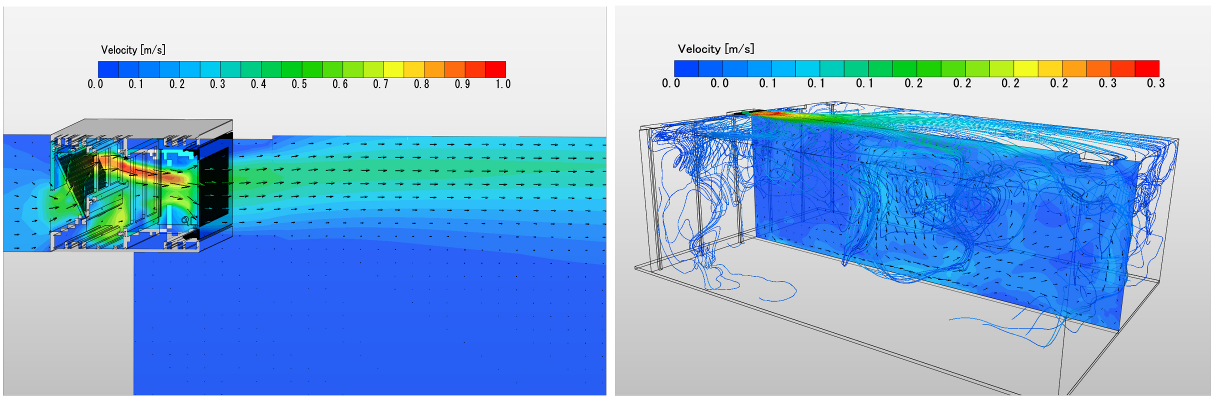

Figure 9: Case study 3 - Trickle ventilator CFD with air intake temperature 10°C warmer than room temperature

Figure 9 shows that the streamlines of the airflows exiting from the trickle ventilators stay close to the ceiling surface. The streamlines for Case study 3 are better dispersed in the living environment than for Case study1.
The "Trickle vent Model" results confirm that intake temperature largely impacts airflow spatial distribution. Three cases were studied only; there is a plan to evaluate the whole temperature range that could be experienced across the four seasons.

3.3 Apartment Model: defining the airflow behaviour and mass/thermal transfer

A two-bedroom 90 m^2 apartment occupied by two adults (male, 40 years old and female, 40 years old) and one child (10 years old) was simulated. The occupants undertake light effort tasks (MET 1.5) for three hours. Following Equation 1, the required airflow rate Q should be not less than 20 L/s. In this case study, the fans are extracting 32 L/s. The two adults are in the living area, and the child is in the office. The CO_2 generation rate (L/s) is constant for each occupant during sitting tasks/light activity, as displayed in Table 1. In this first simple case study, no difference between intake and the room temperature was assumed (Spring or Autumn seasons as reported in Case study 1 - Trickle vent CFD, Figure 7). The wind velocity on the windward side of the building (the north-facing side in our simulation) was assumed to be close to 0 m/s in the vicinity of the 21st-floor selected apartment. The internal doors were kept open in this simulation. The CFD simulation kept the outside CO_2 background level constant (0 ppm). The results will be increased by 400 ppm.

Figure 10 shows the distribution of the CO_2 concentration (ppm) after three hours of occupancy. In the non-occupied rooms (primary bedroom, child's bedroom, and the child's bathroom), the CO_2 concentration (blue scale) is close to the outside level (400 ppm). As expected, the occupied rooms (the living room with the two adults and the office with the child) show a higher level of CO_2. The apartment was ventilated at a nominal rate of 32 L/s, equivalent to around 10.5 L/s per person (assuming three occupants). At a steady-state, considering the apartment as a single zone model (internal doors open) when the net rate of CO_2 generation and the ventilation rate are constant, and the air is well mixed, Equation (2) is obtained:

$$C_s - C_o = N / V_o \quad (2)$$

Where:
C_s = CO_2 concentration in the space (steady state)
C_o = CO_2 concentration in outdoor air = 400 ppm
N = CO_2 generation rate per person = 0.0046 L/s in average for the three occupants (Table 1)
V_o = outdoor airflow rate per person = 10.5 L/s
From Equation (2), $C_s \approx (0.0046/10.5) + 400$ ppm ≈ 438 ppm + 400 ppm ≈ 838 ppm

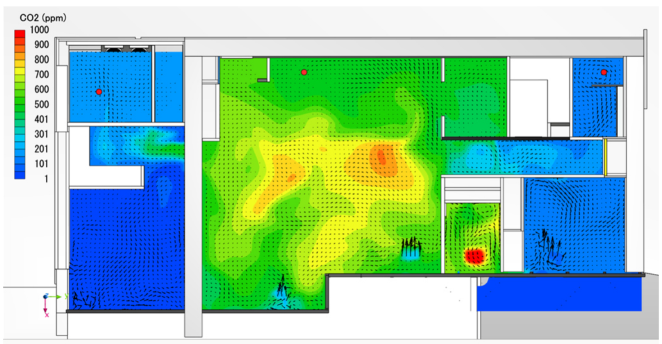

Figure 10: Apartment Model defining airflow behaviour

The results from the CFD simulation (Figure 10) show a CO_2 level in occupied rooms between 700 ppm and 1200 ppm after three-hour occupancy (the closer to the middle of the living room, the higher the CO_2 level). This CFD result slightly exceeds the mass balance estimate for the middle part of the living room. The living room occupants probably receive less than 10.5 L/s of outdoor air from the two trickle ventilators in the living room (6.4 L/s for each ventilator). Decreasing V_o from 10.5 L/s to 6.4 L/s will increase C_s from 838 ppm to 1119 ppm.

4 CONCLUSIONS

This project reports on investigating suitable ventilation solutions within the proposed NCC 2022 requirements using a CFD modelling approach in a Melbourne apartment case study.
A cascade model was designed consisting of a three-step process ("Wind Model", "Trickle vent Model" and "Apartment Model"). The three models are not yet connected at this project stage but have been tested independently.
The preliminary results showed that the facade (balcony) design impacted wind speed values that will be used as an intake for the "Trickle vent Model".
The "Trickle vent Models" showed that the temperature gradient between the intake air and the room temperature impacted the streamlines of the airflows exiting from the trickle ventilators. With no temperature difference between outside air and room air (Case study 1), the streamlines stay close and parallel to the ceiling surface, creating a bypass (not well-mixed air). The winter season result (Case study 2) could potentially generate thermal discomfort close to the window. This result will inform recommendations (heating and cooling locations, door undercuts, and passive temperature control by the trickle ventilator) to reduce the risk of thermal discomfort. It will be interesting to investigate the air distribution with internal doors closed. The results showed some local effects, which depend on temperature gradients, and source locations in the airflow. In the future, the age of air and other performance indicators to estimate the air change effectiveness will be investigated. This project will inform suitable ventilation solutions for people working for extended periods at home (COVID pandemic).
This CFD modelling approach will be validated with indoor air quality (IAQ) sensor deployment in the selected apartment.

5 REFERENCES

Abe, K., Kondoh, T., Nagano, Y. (1994). A new turbulence model for predicting fluid flow and heat transfer in separating and reattaching flows - I. flow field calculations., *International Journal of Heat Mass Transfer*, 37, 139-151.

Bassett, M. (2001) Naturally Ventilated Houses in New Zealand - Simplified Air infiltration Prediction, *CIB World Building Congress, Wellington, New Zealand.*

Hassan, Y. (2017). *An overview of computational fluid dynamics and nuclear applications*, in Francesco D'Auria (Eds), Thermal-hydraulics of water cooled nuclear reactors, Woodhead Publishing, Pages 729-829, ISBN 9780081006627.

Iowa State University (2022, July 12). Iowa Environmental Mesonet.
mesonet.agron.iastate.edu/sites/windrose.phtml?station=YMEN&network=AU__ASOS

Lu, J., Zhu S., Kim M. K., Srebric J. (2019). A Review of CFD Analysis Methods for Personalized Ventilation (PV). Indoor Built Environments*, Sustainability*, 11, 4166.

McNeil, S., Plagmann, M., McDowall, P., Bassett, M. (2015). *The role of ventilation in managing moisture inside New Zealand homes.* BRANZ Study Report SR341. Judgeford, New Zealand: BRANZ Ltd.

Persily, A. et al. (2017). Carbon dioxide generation rates for building occupants. *Indoor Air*, 27(5): 868–879.

Persily, A. (2020). Quit Blaming ASHRAE Standard 62.1 for 1000 ppm CO_2, *in the 16th Conference of the International Society of Indoor Air Quality & Climate (Indoor Air 2020), Seoul, Korea.*

Sørensen, D.N., Nielsen, P.V. (2003). Quality control of computational fluid dynamics in indoor Environments, *Indoor Air*; 13: 2–17.

Stevanovic, Z., Mumovic, D., Kavgic, M. (2009). *Indoor Air Quality and Ventilation Modelling* in Mumovic, D., & Santamouris, M. (Eds.). A handbook of sustainable building design and engineering: An integrated approach to energy, health and operational performance (1st ed.). Routledge.

Versteeg, H. K., Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method. Harlow, Essex, England; Pearson Education Ltd.

Yakhot, V., Orszag, S.A., (1986). Renormalisation group analysis of turbulence. I. Basic theory. *Journal of Scientific Computing*, 1, 3–51.