BUILDING ENERGY AND CFD SIMULATION TO VERIFY THERMAL COMFORT IN UNDER FLOOR AIR DISTRIBUTION (UFAD) DESIGN

Matthew Webb[1]
[1] Umow Lai Consulting Engineers, South Yarra, Victoria, Australia

ABSTRACT

Corporate tenants require ever-greater design certainty with respect to all aspects of proposed developments. Because of this, its relative novelty and a design methodology that differs from ceiling-based Variable Air Volume (VAV) air conditioning, Under Floor Air Distribution (UFAD) has faced significant scrutiny. Building simulation offers methods to understand the implications of design decisions. Here, Building Energy Simulation (BES) and Computational Fluid Dynamics (CFD) are used to estimate and verify that the UFAD design for an office building in Melbourne, Australia, can provide sufficient cooling such that the Predicted Mean Vote does not exceed 0.5. Both BES and CFD indicate that the proposed UFAD system would provide sufficient cooling during summer design conditions.

Keywords: EnergyPlus, Computational Fluid Dynamics, thermal comfort, cooling design, Under Floor Air Conditioning.

INTRODUCTION

Corporate tenants involved in large office developments require ever-greater design certainty with respect to all aspects of the proposed development, such as operational energy consumption, daylighting and artificial lighting performance, acoustics and occupant comfort. Further, Under Floor Air Distribution (UFAD) is becoming more popular in these types of projects due to purported improvements in energy efficiency and Indoor Environment Quality (IEQ) over ceiling-based Variable Air Volume (VAV) air conditioning (which is the more conventional approach in Australia). However, because of client demands to obviate risks, its relative novelty and a design methodology that differs significantly from VAV, UFAD has faced significant scrutiny in high profile projects.

Building simulation offers methods to scrutinise and understand the implications of design decisions on the final construction. In the case of UFAD, it allows building services designers a greater level of confidence that an underfloor system can meet the required comfort conditions, i.e. a particular range of Predicted Mean Vote (PMV) or Percentage People Dissatisfied (PPD). Both Building Energy Simulation (BES) and Computational Fluid Dynamics (CFD) can be used in this process.

This study focuses on the multi-storey office project at 150 Collins St, Melbourne, Australia. Here, the use of BES software and CFD analysis was important to verify the proposed UFAD design and provide confidence to the client that the solution would function effectively, especially under sunny summer conditions. The rationale for the study was to verify the proposed UFAD design via BES and CFD. The analysis also allowed a detailed comparison of results from these two forms of building simulation.

BACKGROUND TO BUILDING ENERGY SIMULATION

As described by Crawley et. al. (2005, 2008), building energy programs been developed for the past 50 years, leading to a multitude of specialities, methods and end-user programs. Crawley et. al. survey “twenty major building energy simulation programs” (2005, p. 2). It is clear from this assessment that the range of available programs and features is broad.

In this study, DesignBuilder was used as the primary modelling tool. DesignBuilder offers a rapid model-building OpenGL interface (*DesignBuilder Simulation* 2010) combined with a “full-featured user interface to EnergyPlus HVAC” (*EnergyPlus Energy Simulation Software* 2012). The key to DesignBuilder’s simulation capabilities is the integration of the EnergyPlus calculation engine (*DesignBuilder Simulation* 2010). Critically for this analysis, DesignBuilder also offers an integrated CFD package (*EnergyPlus Energy Simulation Software* 2012).

EnergyPlus, released by the US Department of Energy, is “an energy analysis and thermal load simulation program” with it’s origins leading back to the BLAST and DOE-2 programs (*Getting Started with EnergyPlus* 2012, p1). The EnergyPlus framework involves two major elements: the heat and mass balance simulation and the building systems simulation, controlled by the Simulation Manager

(*Getting Started with EnergyPlus* 2012, p4). The calculation of zone demand relies on a heat balance method similar to that in the *2009 ASHRAE Handbook - Fundamentals*.

EnergyPlus offers a flexible range of options for the simulation of both the building skin and the associated HVAC system (Crawley et al. 2005), created via a series of model objects (*Input Output Reference* 2012). Hence the EnergyPlus simulator was capable of accurately modelling both the building fabric and UFAD HVAC system required for this project.

As noted in *Getting Started with EnergyPlus* (p10), "EnergyPlus is a simulation engine: it was designed to be an element … that would include a graphical user interface to describe the building". While several tools provide Graphical User Interfaces (GUIs), DesignBuilder is the preferred GUI-based simulation tool in the present study.

BACKGROUND TO CFD

Anderson (2001, p162, citing Anderson 1995) defines Computational Fluid Dynamics as "the art of replacing the integrals or the partial derivatives … with discretized algebraic forms, which in turn are solved to obtain numbers for the flow field values at discrete points in time and/or space". Air temperature, airflow, convective heat transfer and species dispersion can be calculated (*2009 ASHRAE Handbook - Fundamentals*, p13.1). Alongside the Navier-Stokes equations for continuity, momentum (in three dimensions) and energy (Benson (ed.) 2012), turbulence is described as a variable diffusion coefficient termed turbulent viscosity. Turbulent viscosity is often calculated from turbulent kinetic energy and its dissipation rate. Alongside an equation for contaminant distribution, there are eight coupled differential equations in total (*2009 ASHRAE Handbook - Fundamentals*, p13.2), plus an equation of state (Benson (ed.) 2012).

DesignBuilder models the aforementioned equations using the primitive variable method (*DesignBuilder Simulation + CFD Training Guide* 2009, p162), in which the equations have the general form (*2009 ASHRAE Handbook - Fundamentals*, p13.1):

$$\frac{\partial}{\partial t}(\rho\varphi) + \frac{\partial}{\partial x_j}(\rho U_j \varphi) = \frac{\partial}{\partial x_j}\left(\Gamma_\varphi \frac{\partial \varphi}{\partial x_j}\right) + S_\varphi \quad (1)$$

Here:

- t = time (s),
- ρ = density (kg/m^3),
- φ = transport property (e.g. air velocity, temperature, species concentration, etc),
- x_j = displacement in j direction (m),
- U_j = velocity in j direction (m/s),
- Γ_φ = generalised diffusion coefficient or transport property of fluid flow, and
- S_φ = source or sink.

DesignBuilder solves the set of equations using a finite difference method applied across a finite volume grid. The simulation is steady state, thus presenting a single 'snapshot' in time for every simulation (*DesignBuilder Simulation + CFD Training Guide* 2009, p163).

DesignBuilder allows the modeller to use the same model for CFD that they created for BES. The software automatically generates a finite volume mesh to fit the objects within the selected study domain based on user-preferred grid spacing (*DesignBuilder Simulation + CFD Training Guide* 2009, p164-165). Initially cell boundaries are set according to user dimensions, but, where necessary, adjusts grid spacing to set cell boundaries adjacent to bounding geometry and internal obstructions. Hence a "block-structured grid" is established (Versteeg & Malalasekera 2007, p310) with each block containing rectilinear grid spacing required to incorporate all model geometry. As noted by Versteeg and Malalasekera (2007, p310), block-structured grids are "extremely useful in handling complex geometries that consist of several geometrical sub-components", as is the case with many internal CFD analyses in the built environment. The modeller has the option to adjust the type of spacing within a block. Also, the modeller can select to mesh the block based on a power law relationship, so that the grid mesh can transition more smoothly from small spacing to wide spacing or vice versa (*DesignBuilder Simulation + CFD Training Guide* 2009, p 184).

Boundary conditions within the CFD grid can be defined in two ways: via bounding surfaces and via user-created assemblies. Bounding surfaces can be defined by temperatures (*DesignBuilder Simulation + CFD Training Guide* 2009, p171). Additional surface objects can be defined to represent heat flux and air flow.

Assemblies, drawn by the modeller, represent internal obstructions that will impact fluid flow through the domain (*DesignBuilder Simulation + CFD Training Guide* 2009, p178). Internal assemblies can include internal equipment, lighting and building occupants. Internal assembly boundary conditions can be defined via a temperature or heat flux (*DesignBuilder Simulation + CFD Training Guide* 2009, p179). However, only the convective heat transfer is considered; the radiative component is not considered (*DesignBuilder Help v3.0* n.d.). Diffusers/vents can be added to assembly surfaces where necessary (*DesignBuilder Help v3.0* n.d.).

Two turbulence models are available: either 'k-ε' or 'Constant effective viscosity'. The 'k-ε' representation is part of the Reynolds Averaged Navier Stokes set of models, where velocity terms are replaced by a mean and fluctuating component. This leads to a calculation of viscosity, derived from turbulent kinetic energy (k) and dissipation rate of kinetic energy (ε). 'Constant effective viscosity' can be more numerically stable and less computationally expensive but ultimately less accurate (*DesignBuilder Simulation + CFD Training Guide* 2009, p190).

Modellers have the ability to define a discretisation scheme from 3 options (*DesignBuilder Simulation + CFD Training Guide* 2009, p190):

- Upwind – because of the one-way nature of convection, the finite difference equation for the convective term using the upstream scheme assumes that the value of a dependent variable on the cell interface is equal to the value of the cell at the upstream side of the interface,
- Hybrid – uses an upwind scheme where Reynolds and Peclet numbers (Pe) are high (i.e. when the ratio of bulk motion to diffusion is large) and a central difference scheme otherwise; more computationally expensive than the 'upwind' scheme',
- Power Law – first order scheme, where diffusion is set to zero for Pe > 10 and uses a polynomial expression for flux; this "produces a better results than the hybrid scheme" (Versteeg and Malalasekera 2007, p155-156).

To control surface and near wall modelling, the CFD solver uses "a conventional log law of the wall terms for the treatment of energy and momentum transfer" (Potter, 2013). Thus a profile is developed based on the logarithm of the distance from the wall.

Dependent variable iteration can also be controlled by using false time step or relaxation factor methods. The false time step is normally preferred method and was used in the current study. Finally, values for acceptable termination residuals can be defined (*DesignBuilder Simulation + CFD Training Guide* 2009, p191).

The DesignBuilder CFD module has been compared against the commercial CFD package Phoenics by Northumbria University. This study concluded that, due to low RMS differences between results for the two programs, DesignBuilder CFD is "comparable with a high-exposure specialist CFD package" (*An Inter-program Analysis of Computational Fluid Dynamics Based on PHOENICS and DesignBuilder Software* 2011).

CALCULATION OF THERMAL COMFORT

ASHRAE (*ASHRAE Handbook: Fundamentals* 2009, p9.1) describes the judgement of thermal comfort as a "cognitive process involving many inputs influenced by physical, physiological, and other processes". The Fanger Predicted Mean Vote (PMV), alongside Predicted Percent Dissatisfied (PPD), are widely used numerical and rigourous measures of thermal comfort that relate the major environmental and physiological variables involved (*ASHRAE Handbook: Fundamentals* 2009, p9.17). The multi-equation model (*ASHRAE Handbook: Fundamentals* 2009, p9.17) can be described in the following terms:

$$\text{PMV} = f\left(T_{ai}, RH, v_i, T_{MRT}, clo, met\right) \qquad (2)$$

Here, evaluated at the occupant position:

- T_{ai} = Indoor air temperature (°C)
- T_{MRT} = Mean radiant temperature (°C)
- v_i = Internal air velocity (m/s)
- RH = relative humidity (%)
- clo = clothing value (1 clo = 0.155 m^2K/W)
- met = activity level (metabolic rate, 1 met = 58 W/m^2)

PPD can be calculated from PMV (*ASHRAE Handbook: Fundamentals* 2009, p9.17).

Both the BES and CFD modules of DesignBuilder can calculate Fanger PMV with user input of occupant clothing (clo) and metabolic (met) rate. The BES calculation is via EnergyPlus (*Input Output Reference* 2012, p329).

DESCRIPTION OF THE STUDY

General Setup

The client mandated a design requirement that the proposed UFAD system must meet the prescribed internal conditions (Table 1) during the summer cooling peak design day. The external dry bulb design conditions for comfort (or non-critical processes) in the building's location of Melbourne, Australia are 34.3°C dry bulb and 20.5°C wet bulb (*AIRAH Technical Handbook* 2007). Therefore the BES and CFD simulations within this study were concerned with a single design day only in order to meet the client requirement. An allowance for blinds was made: blinds (10% solar transmissivity) could be lowered if (when) to mitigate incoming solar loads greater than 100 W/m^2.

Table 1
Interior design conditions

DESIGN CONDITION	MINIMUM	MAXIMUM
Dry bulb	20°C	24°C
Relative Humidity	40%	60%
PMV	-0.5	0.5

As this study used dynamic simulation rather than a generic design day calculation, the actual external conditions were selected from the weather file. A suitable time, slightly exceeding the required design conditions, was found to be February 7, 16:00 (shown in Table 2).

Table 2
External design conditions

DESIGN CONDITION	VALUE
Dry bulb	34.5°C
Wet Bulb	23.5°C

DESIGN CONDITION	VALUE
Global direct solar	935 Wh/m^2

To test the HVAC system's performance under these conditions, an accurate model of the building architecture was constructed in DesignBuilder. This is illustrated in Figure 1 below.

The building is 12 storeys tall, with Levels 5-12 having a common envelope. Levels 5-12 also have a common HVAC zoning arrangement. From a peak thermal comfort perspective, there is no significant shading. Thus Levels 6-11 are assumed to be thermally equivalent. and Level 8 was selected as the common template.

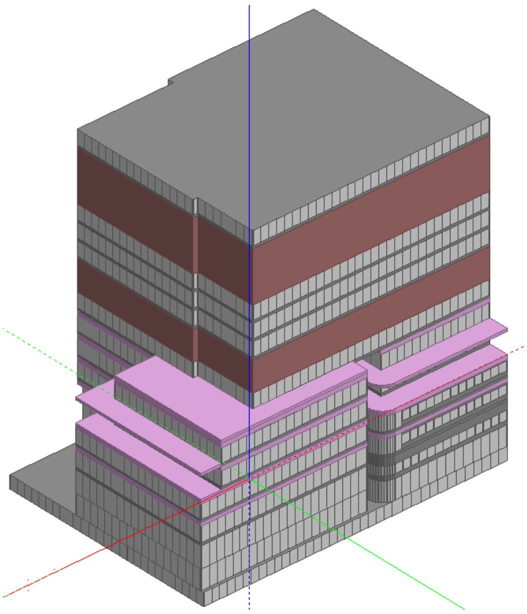


Figure 1 DesignBuilder model

Building envelope details, including glazing, spandrel panels and wall constructions were input into the DesignBuilder model. HVAC zones were emulated by inserting partitions. Supply plenums and equivalent zones were also created beneath each occupied floor. The general upper tower layout (specifically Level 5) is shown in Figure 2.

The aim of the study was to test the thermal comfort of the UFAD design under peak design conditions. To this end, Level 8 was selected as exemplary and was the focus of the study. To maintain thermal accuracy, Levels 7 and 9 were created below and above Level 8, rather than treat the floor and ceiling as adiabatic.

North and west facades would be exposed to the greatest solar loads at the hottest period of the summer design day and were thus selected for the detailed assessment. As the client requested testing very close to the façade, additional divisions were inserted into the north and west perimeter zones on Level 8. The partitions created additional façade zones 1.0m wide. A 3.0m×3.0m cubicle office was added on the northwest corner. This office would serve as an example office in the BES and form the basis of the CFD model. The Level 8 partition plan is shown in Figure 3.

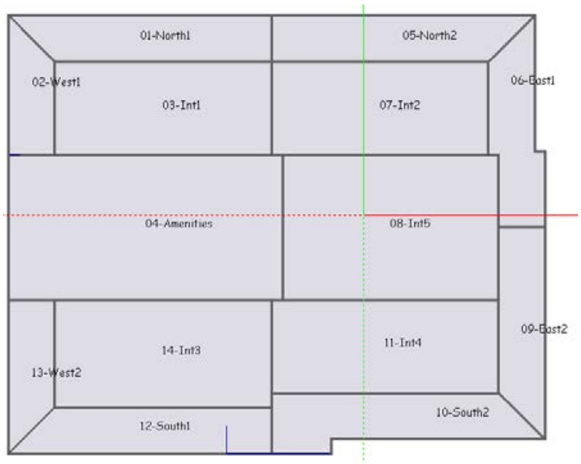


Figure 2 Level 5 Layout

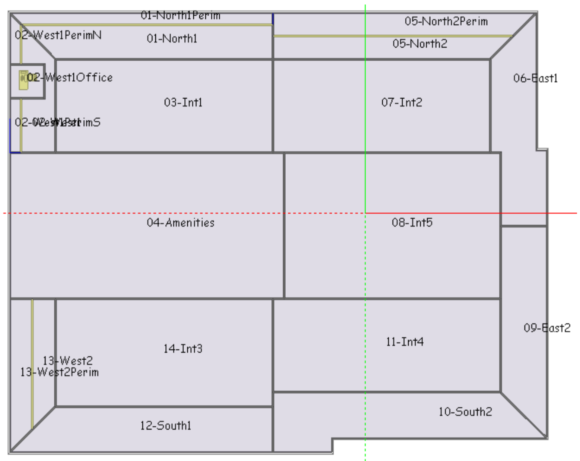


Figure 3 Level 8 Layout with additional partitions

Building Energy Model

The HVAC plant and air distribution model was created in DesignBuilder using the 'Detailed HVAC' module (*DesignBuilder Help v3.0* n.d.).

The 150 Collins St HVAC design consists of centralised chiller and boiler sets supplying chilled and heating hot water to Air Handling Units (AHUs) and reheat coils. The AHUs in turn supply cooling and heating air to underfloor plenums before air diffuses into the occupied zones. Supply plenum zones are designated as such in the model. Separate air conditioning (not considered in this study) is designed for specific spaces such as the ground floor lobby and fan coil units for Level 2 function rooms.

EnergyPlus offers a number of "Room Air Models" to simulate air distribution models that differ from a standard fully mixed air condition (*Input Output Reference* 2012, p302). For this project the Three Node Displacement Ventilation model was implemented in a detailed EnergyPlus model (*Input Output Reference* 2012, p313). This model provided a more realistic representation of internal air conditions.

The HVAC system was scheduled to operate as per the design brief of 55 hours per week, 07:30-18:30 Monday to Friday.

Internal loads were set as per the design brief:

- Occupancy – 1 person per 10m^2,
- Lighting – 10 W/m^2,
- Internal equipment – 15 W/m^2.

Outside air was set at 18.75 L/s/m^2 in line with the goal to achieve 3 points under the IEQ-1 Ventilation Rates credit as part of the Green Star Office Rating (*Green Star Technical Manual* 2008, p43). Internal loads were diversified as per the standard NABERS modelling schedules (*NABERS Guide to Building Energy Estimation* 2011). Occupant metabolic rate was set at 1.05 met, equivalent to light office work. Clothing was set at 0.6 clo.

CFD Model

In parallel with the BES, DesignBuilder was employed to create a CFD model (see Figure 4). This model was created in the west-facing Level 8 office zone that was created as part of the specific zone partitioning (see above).

Internal obstructions and loads were added to the model to reflect the design brief and create an equivalent scenario to the BES. A desk, seated occupant, computer and ceiling light were added to represent internal loads.

With reference to the background section above, the following calculation parameters were selected for the CFD simulation:

- The k-ε turbulence model was used,
- The Power Law discretisation method was selected,
- Maximum mesh spacing of 0.05m (with DesignBuilder creating block-structured grid to cater for all geometry),
- Maximum dependent variable residual (iterative convergence error) of 10^{-6}.

Due to the small cell spacing, the overall mesh did not require further editing or smoothing. Feedback from DesignBuilder showed an acceptable grid mesh in terms of the cell aspect ratio (the maximum allowable is 50) and memory required for simulation.

To specify boundary conditions, the EnergyPlus simulation was run over the summer period. The following simulation results on the design day (February 7) were used as CFD boundary conditions:

- External façade temperature,
- Internal partitions temperatures,
- Supply air quantity and temperature,
- External solar façade load.

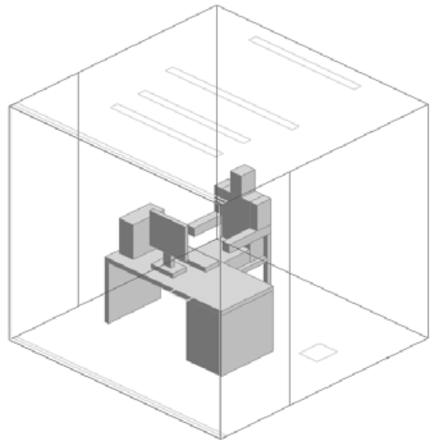


Figure 4 CFD Office

SIMULATIONS

The BES was run over the period 27 January – 10 February with a calculation frequency of 15 iterations per hour. The simulation was started in January to allow for sufficient warm up and stabilisation.

Following execution of the BES, surface, air and internal loading boundary conditions were extracted for the CFD model, as shown in Table 3.

Table 3
CFD Boundary Conditions

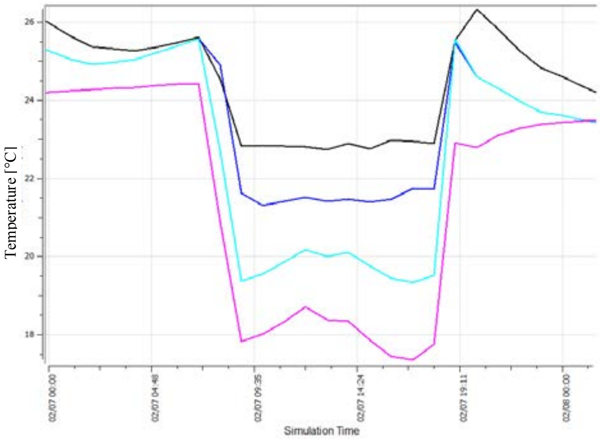
INPUT/OUTPUT	VALUE
Temperatures [°C]	
External temperature	35.0
Initial air temperature	22.5
Floor temperature	23.4
Ceiling temperature	24.2
Internal partition temperature	23.4
External wall inside surface temperature	23.9
External window inside surface temperature	29.2
Supply air	
Supply temperature	17°C
Airflow supply rate	100 L/s
Load inputs [W]	
Lights	90
Equipment	150
Occupancy (sensible/convective)	69.4

RESULTS

Building Energy Simulation

The BES provided an accurate estimation of the operational performance of the UFAD system across Level 8. Figure 5 shows the stratification of the air temperatures in the West Office (the basis for the CFD model). This clearly illustrates the stratification of air that develops in the Three Node Displacement Ventilation Room Air Model that was implemented from EnergyPlus.

The Mean Radiant Temperature for the office is plotted in Figure 6. The MRT at the time of the CFD simulation (16:00) is 24.4°C. Table 4 indicates the thermal comfort in the north and west zones.



LEGEND	
	Supply air temperature from plenum to thermal zone [°C]
	'Floor' zone temperature [°C]
	'Occupied' zone temperature [°C]
	Upper 'mixed' zone temperature [°C]

Figure 5 West Office internal air temperatures

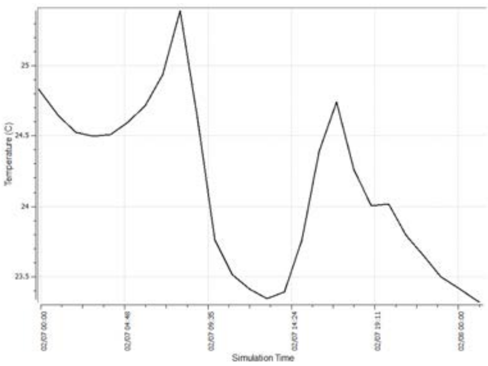


Figure 6 MRT in West Office

Table 4
Thermal Comfort Results from BES

ZONE	AIR TEMP [°C]	PMV
North Zones		
North 1	21.0	-1.50
North 1 (Perimeter)	21.5	-1.37
North 2	21.1	-1.59
North 2 (Perimeter)	21.2	-1.45
West Zones		
West 1	21.0	-1.61
West 1 North (Perimeter)	22.6	-0.72
West 1 South (Perimeter)	22.6	-0.66
West Office (CFD model)	21.5	-1.51

Computational Fluid Dynamics

Following the CFD simulation, slices were plotted for the key thermal comfort outputs. Air temperature and velocity are shown below. The plan view plots have been set at a height of 1.2m, equal to the thermal comfort and thermostat height from the BES. The scale for the temperature and velocity plots is shown in Table 5.

Table 5
CFD plot colour scales

COLOUR	TEMP. [°C]	VELOCITY [m/s]	MRT [°C]	PMV
	18.00	0.00	23.40	-2.00
	18.64	0.02	23.45	-1.82
	19.27	0.05	23.51	-1.64
	19.91	0.07	23.56	-1.46
	20.55	0.09	23.62	-1.27
	21.18	0.11	23.67	-1.09
	21.82	0.14	23.73	-0.91
	22.45	0.16	23.78	-0.73
	23.09	0.18	23.84	-0.55
	23.73	0.20	23.89	-0.36
	24.36	0.23	23.94	-0.18
	25.00	0.25	24.00	0.00

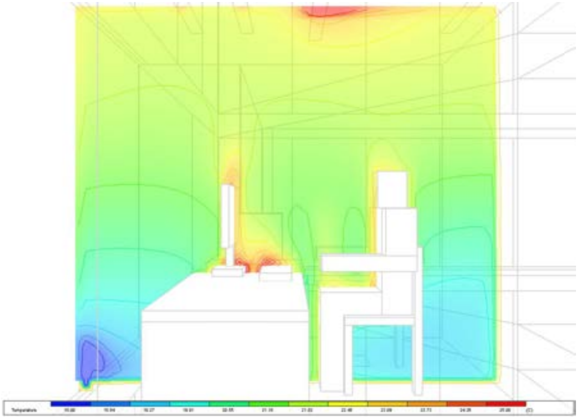


Figure 7 Air Temperature (side view)

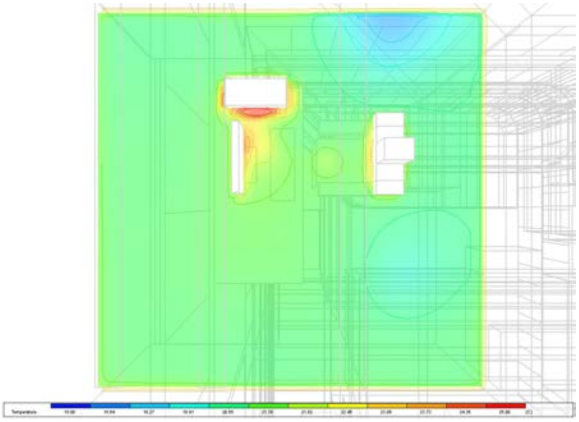


Figure 8 Air Temperature (height 1.2m)

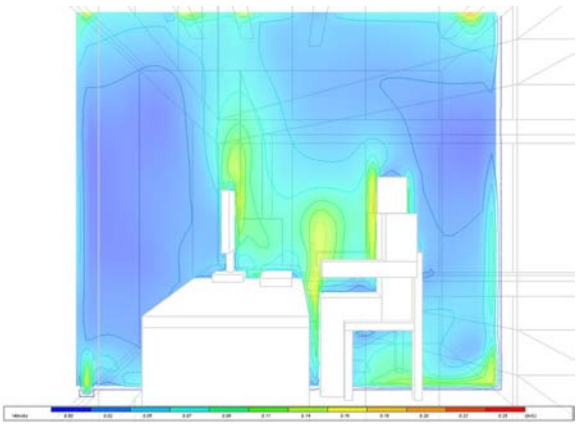


Figure 9 Air Velocity (side view)

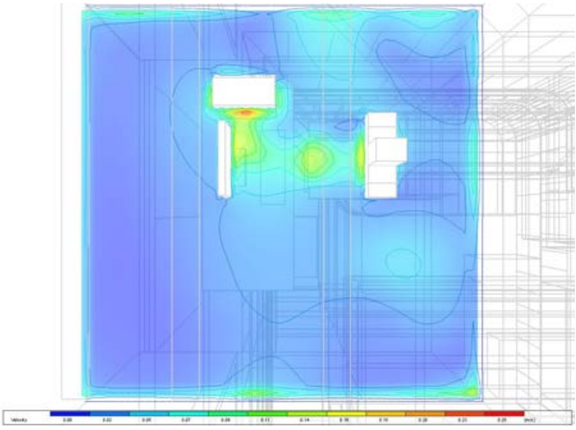


Figure 10 Air Velocity (height 1.2m)

DesignBuilder computes thermal comfort data in a post-CFD process. Mean Radiant Temperature is evaluated across the CFD mesh and then the PMV is calculated. Occupant data is input by the user – in this case 1.05 met and 0.6 clo as per the BES. The scale for MRT and PMV is shown in Table 5.

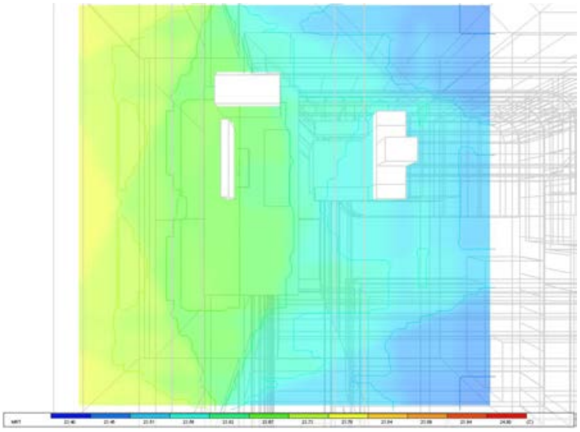


Figure 11 MRT (height 1.2m)

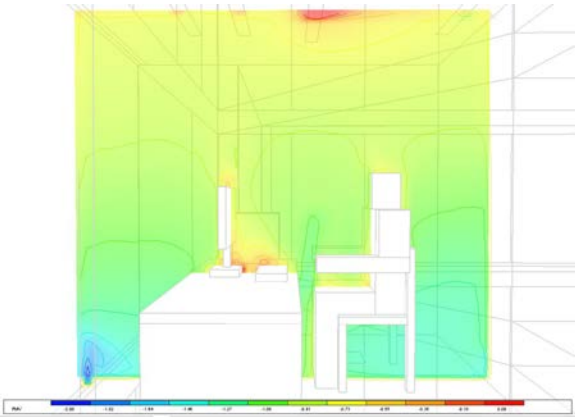


Figure 12 PMV (side view)

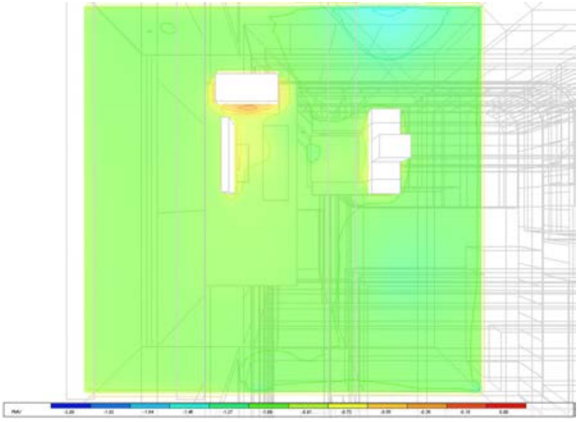


Figure 13 PMV (height 1.2m)

DISCUSSION

The results show a high degree of similarity between the results for the BES and the CFD. In particular, the stratification of air temperatures compared favourably, as seen in Table 6 below.

Table 6
Temperature comparison

STRATIFICATION ZONE	BES [°C]	CFD [°C]
Supply	17.4	17.5
Floor subzone	19.5	19.7
Occupied subzone	21.5	21.8
Mixed subzone	23.0	22.8

For both BES and CFD, the MRT in the occupied zone was almost identical. This was expected as BES surface temperature results form CFD boundary conditions. Figure 11 shows a variation of MRT through the CFD domain, with points closer to the façade showing slightly warmer temperatures. The BES calculated only one MRT per zone based on an area-weighted mean.

We see from Table 4 that PMV (at a height of 1.2m) is -1.51 from the BES. This compares with the PMV from CFD in Figure 13, which indicates an average PMV of approximately -1.2 to -1.1. The small discrepancy here can be accounted for by a difference in velocity for comfort calculations. For the BES, velocity is a user-defined input into DesignBuilder and was set at 0.25 m/s. This contrasts with the CFD result in Figure 10, which shows velocities of approximately 0.1 m/s.

These low thermal comfort predictions may appear counterintuitive, as the air temperature is within the required design range of 20-24°C. However, according to the ASHRAE thermal sensation scale (*2009 ASHRAE Handbook - Fundamentals*, p9.11), a prediction where $-2.0 < PMV < -1.0$ indicates that occupants feel ‘cool’ to ‘slightly cool’. Since the ideal PMV is a value of zero (occupant thermal sensation is ‘neutral’, neither ‘warm’ nor ‘cool’), this result is less than optimal. Still, in terms of a summer design scenario, the simulations indicate that the UFAD HVAC system has adequate capacity to sufficiently cool the test space. Further adjustment of the thermostat controls would allow a decrease in the supply volume or increase in supply temperature, subsequently increasing the zone PMV.

FURTHER WORK AND CONCLUSIONS

DesignBuilder software was used in this case study to develop a detailed architectural model that formed the basis for BES and CFD. To model a UFAD system, the BES model employed the Three Node Displacement Ventilation model available through EnergyPlus. The BES results provided boundary conditions to the CFD model, which gave a more detailed visualisation of comfort variables within the case study office.

From a design perspective, both BES and CFD simulators provide opportunities to test and verify HVAC designs. In this case study, for a central Melbourne office tower, both BES and CFD were able to provide evidence that the proposed UFAD system would maintain adequate occupant thermal comfort during summer design conditions as specified by the client. Indeed, the BES and CFD results were such that the estimated PMV was less than -1.0, signalling that occupants would feel cool rather than too hot. Thus the simulations predicted that the UFAD system had sufficient cooling capacity in the western zone that was tested. In the perimeter office that was tested, the supply air flow rate was approximately 11.1 L/s/m^2, or 13.3 Air Changes per Hour. While this is not unusual for a perimeter zone at summer design conditions, the controls within this office and other perimeter zones could be refined to increase the internal temperatures. This would elevate the PMV closer to the ideal value of zero. Additional BES and CFD simulations would confirm the sensitivity of PMV to marginal adjustments to flow rates and supply air temperatures. However, during commissioning, the controls must be tuned to provide acceptable conditions across multiple zones served by the one air handling unit. Therefore, exact tuning of one individual perimeter office zone is not necessarily a practical option.

Individually, both BES and CFD are able to give detailed feedback on the suitability of HVAC designs to meet client comfort criteria. In this case, both BES and CFD provided equivalent conclusions, and were able to provide added confirmation. Both simulations verified the acceptability of the UFAD design.

REFERENCES

AIRAH Technical Handbook 2007, 4 edn, The Australian Institute of Refrigeration, Air Conditioning and Heating (Inc.) (AIRAH), Melbourne.

An Inter-program Analysis of Computational Fluid Dynamics Based on PHOENICS and DesignBuilder Software 2011, viewed 31-Jan 2013, http://www.designbuilder.co.uk/ component/option,com_docman/task,cat_view/gid,20/Itemid,30/

Anderson, JD 1995, *Computational fluid dynamics: the basics with applications*, McGraw-Hill.

Anderson, JD 2001, *Fundamentals of aerodynamics*, McGraw-Hill.

ASHRAE Handbook - Fundamentals 2009, ASHRAE, Atlanta, USA.

Benson, T 2012, *Navier-Stokes Equations - 3-Dimensional Unsteady*, http://www.grc.nasa.gov/WWW/k-12/airplane/nseqs.html.

Crawley, DB 2008, 'Contrasting the Capabilities of Building Energy Performance Simulation Programs', *Building and Environment*, vol. 43, no. 4, pp. 661-73.

Crawley, DB, Jon W. Hand, Michaël Kummert & Griffith, BT 2005, 'Contrasting the Capabilities of Building Energy Performance Simulation Programs', United States Department of Energy, University of Strathclyde, and University of Wisconsin, Washington, DC, USA.

DesignBuilder Help v3.0 n.d., viewed 01-Feb 2013, http://www.designbuilder.co.uk/helpv3.0/.

DesignBuilder Simulation + CFD Training Guide 2009, vol. 2013, United Kingdom, viewed 01-Feb 2013, <http://www.designbuilder.co.uk/content/view/106/186/>.

DesignBuilder Simulation 2010, viewed 01-Feb 2013, http://www.designbuilder.co.uk/index.php?option =com_content&task=view&id=7&Itemid=13.

EnergyPlus Energy Simulation Software 2012, viewed 08-Feb 2013, <http://apps1.eere.energy.gov/buildings/energyplus/ep_interfaces.cfm>.

EnergyPlus Engineering Reference, 2012, University of Illinois and the Ernest Orlando Lawrence Berkeley National Laboratory.

Getting Started with EnergyPlus, 2012, University of Illinois and the Ernest Orlando Lawrence Berkeley National Laboratory.

Input Output Reference - The Encyclopedic Reference to EnergyPlus Input and Output, 2012, University of Illinois and the Ernest Orlando Lawrence Berkeley National Laboratory.

NABERS Guide to Building Energy Estimation 2011, NSW Office of Environment and Heritage, viewed 03-Feb 2013, http://www.nabers.gov.au/ public/WebPages/ContentStandard.aspx?id=410 &module=50&templat e=2.

Potter, Steve 2013, Background detail for CFD, viewed 20-Apr-2013, http://www.designbuilder.co.uk/component/option,com_forum/Itemid,25/page,viewtopic/t,4043/sid,7cc1ce077d0844af13ac955237da37ee/*Technical Manual - Green Star Office Design & As Built Version 3* 2008, Green Building Council of Australia.

Versteeg, HHK & Malalasekera, W 2007, *An introduction to computational fluid dynamics: the finite volume method*, Pearson Education Australia.