PREDICITNG THE EFFECTS OF THERMAL MASS IN IT SUITES USING COMPUTATIONAL FLUID DYNAMICS

Joshua Tolley[1], Malcolm Cook[2]
[1,2]Loughborough University, Loughborough, United Kingdom
[1]Email: joshwtolley@gmail.com,

Abstract

Previous research has been conducted in relation to building thermal mass and ventilation strategies for IT suites using Computational Fluid Dynamics (CFD). However, there is a dearth of literature associated with the physical modelling of thermal mass using CFD software. This paper fills this gap in research, using CFD to predict the ventilation performance and thermal comfort effects of using thermal mass in IT suites. Simulations were run using the CFD software package PHOENICS. The application of thermal mass to the IT suite has shown a significant reduction in the indoor temperature of up to 4.9°C (17%). The results of this study provide potential cost saving implications and improvements in optimising occupant thermal comfort.

Introduction

Designers are now being driven more than ever to explore new and innovative ways to reduce cooling loads on buildings with growing environmental concerns combined with pledges from countries to meet strict carbon emission deadlines. The implementation of thermal mass is proving to be an accepted method, it is the CFD modelling of this approach that is studied in this paper.

Literature illustrates that CFD has extensive applications, both within and outside the building industry. The quality of the CFD investigation depends on several factors. Firstly, the physics combined with the numeric equations used in the software (Norton et al., 2007). Secondly the degree of knowledge that the modeler has of this physics. This knowledge provides flexibility to the user (Niu et al., 2005) and contributes similar yet more accurate results than physical experiments. Physical experimentation does allow for precise environmental measurements, however a comprehensive analysis would require large amounts of preparation time and expensive equipment (Cook, 1990). In general, the accuracy of CFD is dependent upon the design of the computational grid and the availability of computational power to solve the equations.

It is possible to store heat in the material of the outer envelope and the interior mass of the building to reduce indoor air temperature swings and cooling load peaks. The material in which this is stored is the construction mass of the building itself, which is referred to as the thermal mass (Balaras, 1996).

Spaces housing IT equipment and occupants (IT suites) suffer from significant internal heat gains which need to be cooled to maintain adequate thermal comfort. There is a dearth of literature that specifically focuses on the environmental performance of these spaces. The application of thermal mass and chilled ceilings offer the potential to cool IT suites and improve the interior thermal comfort levels. Furthermore, Antonopoulos and Koronaki (2000) discovered that by increasing the effective layer thickness of the external walls, the internal temperature could be reduced by as much as 2°C.

The aim of this paper is to model thermal mass within a CFD package, hence predicting the effects on ventilation and thermal comfort. This also includes the modelling of chilled ceilings within an IT suite. It is hoped that the conclusions stated in this paper, can be drawn upon and used in further modelling of more complex geometries. This is the rationale for the research reported in this paper.

Details of the case study

For airflow analysis in this study the software package PHOENICS (Parabolic Hyperbolic Or Elliptic Numerical Integration Code Series) was used (CHAM, 2008a). PHOENICS predicts quantitatively: (a) the flow of air in and around buildings, human beings, process equipment and so on; (b) the associated stresses in the immersed or surrounding solids; and (c) the associated changes of temperature and physical composition (CHAM, 2015).

PHOENICS has been validated for a range of applications; for the investigation of the characteristics of multiple plume interactions (Durrani et al., 2011), the evaluation of five $k - \varepsilon$ based models for their perfromance in predicting natural, forced and mixed convection in rooms (Chen, 1995). The air distribution effectiveness was estimated for stratified air distribution systems in classrooms, offices and auditoria (Jiang and Chen, 2009).

Case Study Geometries and Details

The space considered was an IT suite. The space is land locked with no access to the outside air, therefore the air is replenished by two air conditioning units located on the ceiling, at a constant total mass flow rate calculated (eq.

1) at 2.57 kg/s, to maintain a temperature difference of 20 – 24°C. This flow rate is based on equation 1.

$$Q = \dot{m} \cdot Cp \cdot \Delta T \qquad (1)$$

The space has a floor area of 87.36 m^2 (Figure 1) and a height of 4m. The walls are composed of medium density concrete blocks. The ceiling was a concrete slab.

The base case (Figure 2) was designed to show the ventilation effects when the room was at full capacity, with all electronic appliances being used. The space contains 40 computers, positioned in four rows of ten computers, with one extra computer located on the podium used by staff. All the 40 computers positioned in rows have a student sitting at each of them. Six ceiling lights in the space illuminate the enclosed room, providing a sensible heat gain of 54 watts each. A printer and projector are both turned on and at full capacity in this case. Further internal heat gains can be seen in Table 2.

In case 2, chilled ceiling panels are added to the room. The panels cover the entire ceiling surface, represented by the ceiling plate. The temperature of the chilled ceiling plate is a constant 18°C.

In case 3, thermal mass is incorporated into the space. The surface heat flux from the ceiling and wall panels, is reduced to zero Watts. This represents the panels absorbing all the radiative heat gain in the room. This change is to determine whether the application of thermal mass affects the airflow in the space.
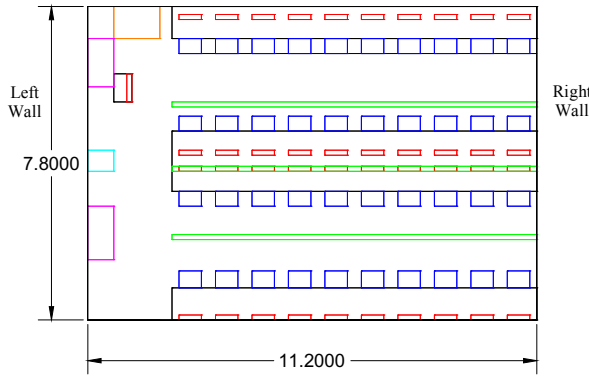

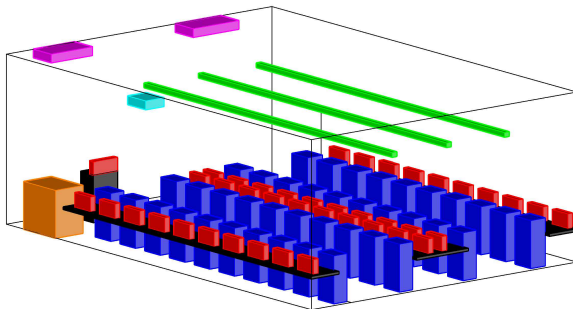*Figure 1: Plan view (Y-direction) of Base Case, with perimeter dimensions (m).*

Figure 2: Isometric view of Base Case. The occupants (blue), computers (red), furniture such as tables and the

podium (black), printer/copier (orange), HVAC units (pink), projector (light blue) and lights (green) can be located in this figure.

Table 1 shows the specifications for each case. The specification states for each case: the total mass flow rate (kg/s): $\dot{m}$; the ambient temperature (°C): T_A, the supply/outlet temperature (°C): T_S; the total convective heat gain in the room (Watts): Q_C; the total radiative heat gain in the room (Watts): Q_R; and the chilled ceiling panel temperature (°C) (if applicable): CC_T.

Table 1: Case specifications.

Case	$\dot{m}$	T_A	T_S	Q_C	Q_R	CC_T
Base					3916.7	n/a
2	2.57	20	20	6297.3	593.4	18
3					0	n/a

Turbulence Model

Different and complex geometries and configurations often make the task of selecting a turbulence model arduous. Conducting simulations with two different turbulence models, to compare the results, was proposed by Nielsen et al. (2007). Similarly, Chen (1995) studied the application of five k-e based models, including the standard $k - \varepsilon$ model (Launder and Spalding, 1974) and the RNG $k - \varepsilon$ model (Yakhot and Orszag, 1986), to compare the results and to recommend a suitable model. Chen (1995) found the RNG $k - \varepsilon$ model to be more accurate for capturing secondary recirculation zones due to its inherent anisotropic modelling assumption, thus stating a recommendation for use in simulations of indoor air flow. Gebremedhin and Wu (2003) also evaluated five different turbulence models in order to determine the appropriate model for a ventilated space. It was seen that the RNG $k - \varepsilon$ model was the most appropriate model, based upon convergence and computational stability criteria. Hussain et al. (2012) evaluated the performance of various RANS turbulence models, for the suitability of prediction of indoor airflow and temperature distributions. The models evaluated included, but were not limited to, the RNG $k - \varepsilon$ model and the SST $k - \omega$ model (Menter, 1992). However, in this study the results showed a relatively better prediction capability of the indoor environment, when the SST $k - \omega$ model was used over the RNG $k - \varepsilon$ model. Cook and Lomas (1998) investigated and compared the performance of the standard $k - \varepsilon$ model and the RNG $k - \varepsilon$ model for predicting buoyancy-driven natural ventilation. Both gave qualitativley similar results, however the theroretical values for the height of the interface were much closer to the predited quantitiative results when the RNG $k - \varepsilon$ model was used.

Most of these comparative studies conclude that the RNG $k - \varepsilon$ model is marginally better than the other models in terms of overall simulation performance when modelling indoor airflow. Therefore, the RNG $k - \varepsilon$ model was selected for use in this study.

Internal Heat Gains

The internal heat gains were split into their relevant convective and radiative elements (CIBSE, 2007), as shown in table 2. The total convective and radiative heat gain for the IT suite was 6372.3W and 3971.7W, respectively. The total radiative heat gain was distributed along the ceiling and wall panels. The ceiling panel covered the entire ceiling of the IT suite with an area of 87.36 m^2. The ceiling and walls accounted for 84.8% and 15.2%, respectively, of their combined total area.

Table 2: Internal heat gains (B – Blockage, P – Plate).

Object	Type	Quantity	Heat Gain		
			Convective (W)	Radiative (W)	Total (W)
Person	B	40	75	55	140
Computer	B	41	52	28	80
Light	B	6	37.8	16.2	54
Photocopier	B	1	737	363	1100
Projector	B	1	201.5	108.5	310
Ceiling	P	1	3323.3		
Wall Left	P	1	296.7		
Wall Right	P	1	296.7		

Thermal Mass

Thermal mass was modelled in the PHOENICS VR using three objects in the IT suite. The three objects covered the entire ceiling and half of the left and right side wall. The objects were modelled as plates with a constant heat flux equivalent to the total radiative heat gain. The radiative heat gain was spread equally over the three plates according to the total area. In the CFD cases in which thermal mass was used, the constant heat flux radiated from the plates was zero, to simulate the ceiling and walls absorbing the total radiative heat gain. This was similar to a high thermal mass material in which the heat would be absorbed and released during the cooler hours of the night. The thermal mass structure is medium density concrete blockwork with a thickness of zero, to allow the wallas to be simply modelled as plates in PHOENICS. Relevant U-values were not needed in order to create this thermal mass in PHOENICS. No exposed thermal mass was present on the floor of the space.

Evaluation Criteria

The target temperature for the space was 19°C to 24°C (CIBSE, 2007).

The PMV (Predicted Mean Vote) is an index defined in ISO 7730 that provides an indicator of thermal comfort on a 7-point thermal sensation scale. The following input parameters were used to calculate the PMV: the radiant temperature was set as 25°C, the clothing insulation was set to 0.6 clo; the practical rnage is between 0 clo (no clothing) and 4 clo (Eskimo clothing); 1lb (0.454kg) corresponds to roughly 0.15 clo, with 0.6 clo and 1.0 clo being the typical summer and winter clothing respectively (CHAM, 2008b). The metabolic rate, measured in 'met' (metabolic units) or W/m^2; 1 met = 58.15 W/m^2, was set to sedentary activity (office, dwelling, school, laboratory): 1.2 met, the external work was set to 0 met. Inidividual comfort is influences by the humidity of the air, which affects the heat loss through the skin, the relative humidity was set to 50%. An acceptable range for the PMV, of between 0.7 and -0.7 (Nicol and Wilson, 2010) was used in this study. PMV was calculated by PHOENICS.

Details of the computer model

The domain's initial temperature was set to the ambient temperature of 20°C, the ambient pressure was 0 Pa, relative to reference pressure of 101325.0 Pa. The fluid properties at ambient temperature were calculated by PHOENICS based on the ambient temperature. The physical fluid properties were as follows; conductivity (κ) set to 0.0257 W/m^{-1}K^{-1}, density (ρ) set to 1.225 kg.m^{-3}, acceleration due to gravity (g) set to -9.81 m.s^{-2}, specific heat capacity (Cp) set to 1004 J.kg^{-1}K^{-1} and thermal expansion (β) set to 0.00333 K^{-1}.

Virtual Model

The occupants were simulated as 'BLOCKAGEs' with a constant heat flux equivalent to the convective fraction of the sensible heat generated by the student. The six suspended ceiling lights, 41 computers and 40 occupants, were simulated as individual BLOCKAGE items, with a constant heat flux equivalent to the convective fraction of the sensible heat generated by the light. The three tables that the computers sat upon were simulated as adiabatic BLOCKAGE objects. A BLOCKAGE item, in PHOENICS, is a volume object that may stop flow within itself, any region occupied by a BLOCKAGE does not exist in the calculations (CHAM, 2016). A small opening, with dimensions 1.06m by 0.87m, was located in the centre of the ceiling panel.

Buoyancy was modelled using the Boussinesq approximation in which density is assumed to be constant, except for a source term in the momentum equation. This is applicable when temperature differences are small, such as those which occur in building air flows. The

quadratic loss coefficient (Q_L) for the opening was set to 2.69, equivalent to a discharge coefficient of 0.61.

Inlets, from the HVAC (Heating ventilation and air conditioning) units, were set up to provide fresh cool air to the room (hereafter referred to as the HVAC's "outlet") at a mass flow rate of 2.57 kg/s and temperature of 20°C. Short simulations were run to test the objects side, either high or low. The object side was set to 'high' for the HVAC's outlets. Further inlets were created to remove air from the domain (hereafter referred to as the HVAC's "intake") at a mass flow rate negatively equal to the outlet: -2.57 kg/s. This negative value assured that the intake would remove air from the room. The object side for the HVAC's intake's were set to 'low'.

Mesh Generation, Selection and Final Refinement

Yao et al. (2012) utilised meshing as part of their CFD analysis, they explain that it is vital to look at different meshing techniques to assess the optimal grid quality that results in a feasible computing time to achieve the precision necessary.

A preliminary study was carried out, in which the grid was systematically refined to examine the numerical accuracy, compared to the simulation time. A tradeoff between these two parameters should be found, such that the simulation duration is reasonable, yet still provides adequate numerical accuracy from the CFD simulation. Fine (Mesh A), medium (Mesh B) and coarse (Mesh C) meshes were developed in PHOENICS. The different meshes used in the mesh dependency study can be seen in Table 3.

Table 3: Meshes used in the mesh dependency study.

Mesh	Structure	X	Y	Z	No. of cells
A	Automatically generated by PHOENICS	129	48	127	786384
B	Mesh A structure but coarser	92	37	78	265512
C	Mesh B structure but coarser	66	29	51	97614
D	Refined from Mesh B	94	44	83	343288

The quantitative results showed a relatively small difference between Mesh B and C. The more detailed mesh, Mesh B, gave a lower reading of velocity average. This suggested that the true velocity of the air in the domain was much closer to the value of Mesh B than Mesh C, as the numerical accuracy will increase simultaneously with the more detailed mesh.

For the qualitative analysis of the results, the three meshes were analysed using the contoured profiles given in the post-processor stage of PHOENICS. The first mesh, Mesh A, was seen to have an unreasonable duration (195 hours) in order to achieve convergence. This suggested that the mesh was too detailed to be completed in a reasonable duration. The results showed that Mesh B analysed the domain in more detail, when the mesh had more cells. The results also showed a more detailed prediction of the PMV around the locations where the occupants would sit. For these reasons, Mesh B was selected to be further refined using different techniques in PHOENICS, due to the trade off between reasonable simulation duration and high numerical accuracy.

In summary, Mesh D was created, from the refinement of Mesh B, using many tactics in PHOENICS geometry settings to create a mesh that was numerically accurate yet still had a reasonable simulation duration time. After running simulations to try to substantiate that Mesh D was satisfactory to use in further cases, convergence was found difficult to achieve. The belief that the mesh was too complex for default relaxation settings in PHOENICS was proposed. Thus, relaxation control in the form of false time-steps was added to the momentum equations (0.1s) and the energy equation (100s). Subsequently, a reduction in the residuals for each equation and stability of the values at the monitoring point gave reason to believe that convergence was achieved at a maximum number of iterations of 10000 (Figure 3). Thus, results for the Base Case were produced (Figures 5,6,7 and 9).
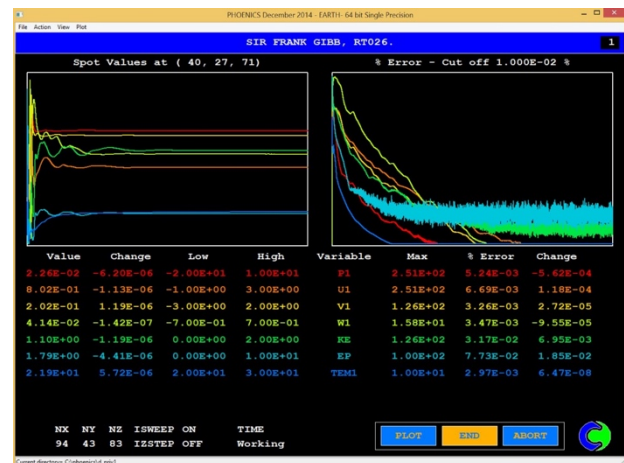

Figure 3: Convergence achieved for Base Case.

Results and discussions

All Cases were simulated in PHOENICS, to provide results at the slices (Figure 4). The maximum number of iterations used to produce the results for the Base Case was 5000. The maximum was reduced to 2500 for subsequent cases. This reduction still provided convergence and high numerical accuracy, yet the major benefit was the vast decrease in simulation duration.

The legends in the figures used to show results fro PHOENCIS, are different as this allow the gathering of

more in depth observations, including where there are peaks and troughs in the temperature and the PMV. Also the arrangement of the temperature and PMV within the domain can be viewed in higher detail, than if all figures had the same legends.
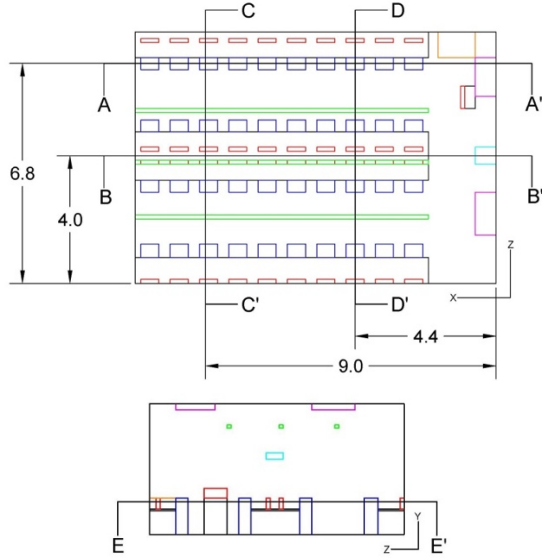

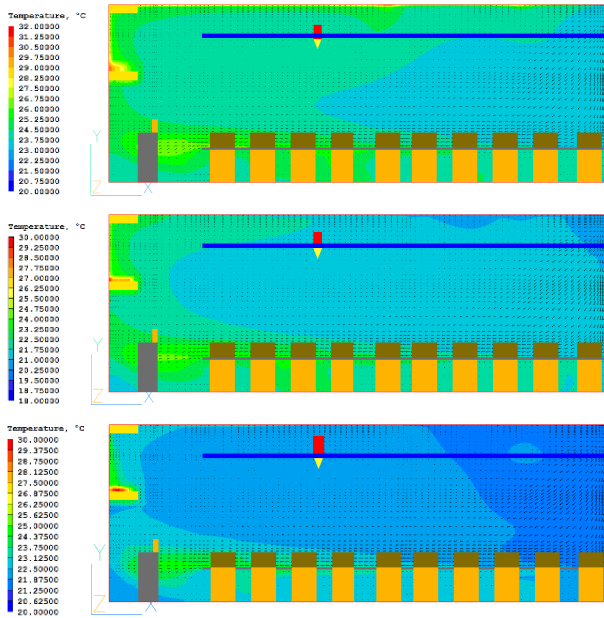Figure 4: Slice Locations: A-A', B-B', C-C, D-D', E-E' (y=1.0m).

Temperature

Figure 5: Temperature profiles at B-B'; Base Case (top), Case 2 (middle), Case 3 (bottom).

The HVAC outlets, located at the top left of profile B-B' (Figure 5), provide fresh air to the space at a temperature of 20°C. The airflow distribution followed a general trend; as the air left the outlet it propogated along the space towards the right wall (Figure 5), where it then recirculated downwards towards and along the table. This propogating air immediately adjacent to the ceiling in case 1 had a higher temperature than the rest of the space. This was due to the radiative heat gain from the ceiling plate. As the air passed along the table past the computers, the temperature rose. The air reached the end of the table and rose back up towards the inlet's on the underside of the HVAC units, due to the buoyancy of the air.

Profile B-B' (Figure 5) also indicated that the bulk of the domain's temperature fell within the acceptable range. Notice should also be given to the localised increases in the temperature of the air around the projector, located on the left wall (Figure 5), where the temperature rose to 32°C in the Base Case.

Profile D-D' (Figure 6) indicated a large 'mushroom' shaped plume of air, rising between the two right side rows of occupants and diffusing outwards before retracting back into the right side HVAC unit. This is clearly represented in Case 3, by the lighter blue contours (Figure 6). Sharp increases in temperature were also seen in and around the computers and occupants, due to the surface heat flux radiated from the heat gains. The air adjacent to the ceiling in the Base Case was shown to have a slight increase in temperature. This was due to the radiative heat gain emitted from the ceiling plate. In Case 2, the chilled ceiling reduced this slight increase of temperature, due to the air consequently being cooled by the 18°C ceiling plate (Figure 6). In Case 3 the radiative heat gain was absorbed by the ceiling plate. Thus, the entire space in Cases 2 and 3 had a reduced temperature on average of 2.3°C and 1.8°C respectively (Figure 8).
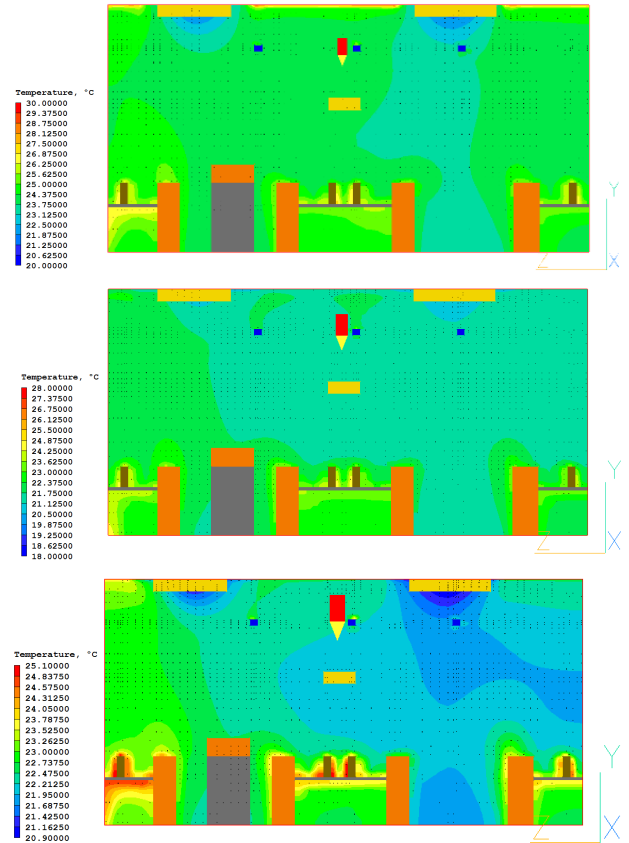
Profile E-E' (Figure 7) denoted the temperature between the rows of occupants was significantly lower than the temperature in the close proximity to the heat gains, especially between the two middle rows of computers, at which the temperature reached its highest. The air flowing from right to left (Figure 7) gradually increased in temperature, due to the surface heat flux from the heat gains. As the air flowed along past the computers and occupants, the air increased in temperature and caused trails of heat (red contours, Case 3 Figure 7), just downwind of the computers.

As seen with the other profiles, the addition of thermal mass or a chilled ceiling to the IT suite considerably reduced the mean temperature of the space. However, unlike profiles B-B' and D-D' where the lowest temperature was found in Case 2, profile E-E' indicated the lowest temperature was present in Case 3.

The Base Case average temperature was significantly higher than both cases 2 and 3 (Figure 8). The different slice locations showed Case 2 had the lowest temperature for slices B-B' and D-D', however Case 3 had the lowest temperature in the E-E' slice (Figure 8). Case 2, with the chilled ceiling panels present, had the lowest overall temperature of 22.93°C; a decrease of 2.61°C from the Base Case. Furthermore, the majority of the profiles demonstrated that the implementation of thermal mass or a chilled ceiling, produced temperatures that fell within the acceptable range of 19-24°C.
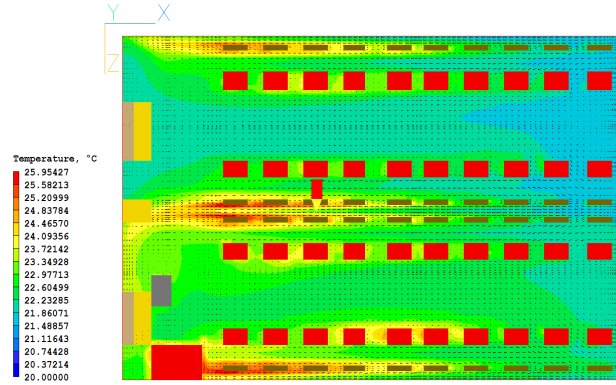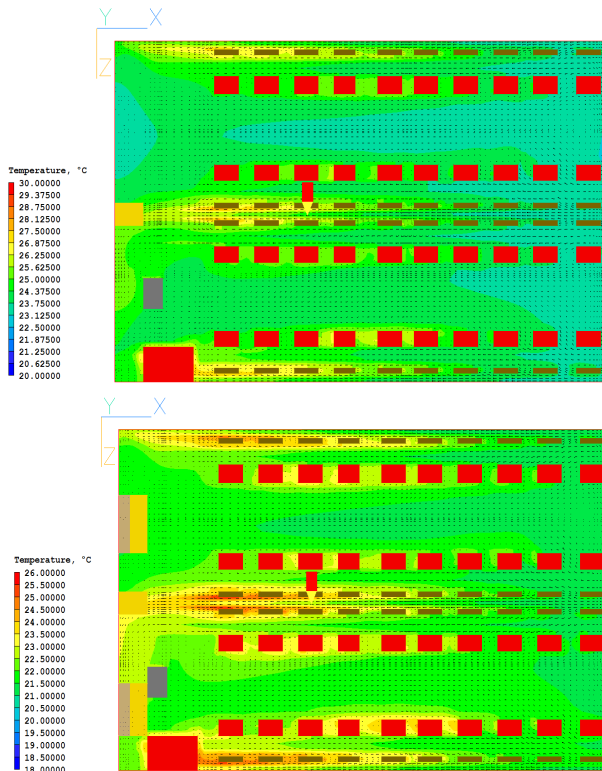

Figure 7: Temperature profiles at E-E'; Base Case (top), Case 2 (middle), Case 3 (bottom).

The concensus was that the effect of employing a chilled ceiling or thermal mass considerably reduced the temperature of the space, by a maximum of 2.65°C and 4.91°C respectively.
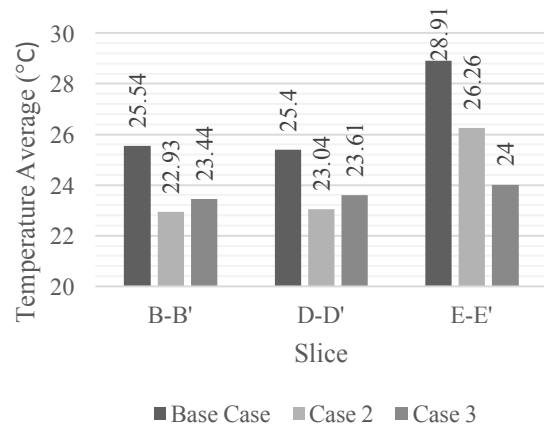

Figure 8: Temperature average at slices B-B', D-D' and E-E'; for base case, case 2 and 3.

Predicted Mean Vote

The profile E-E' (Figure 9) provided a very clear picture of the thermal sensation of the occupants in the IT suite with the application of a chilled ceiling panel or thermal mass.

The air flowing in between the rows of occupants had a vastly reduced PMV value, meaning that the thermal sensation was significantly cooler at these locations. The air in these locations was not obstructed or heated, hence it flowed freely from right to left (Figure 9). Areas of higher PMV values were located to the left of the heat gains as the air flowed adjacently and was heated (Orange and yellow trails, Figure 9).

Due to the heat gains within the space, specifically the occupants and computers, the PMV increased to above 0, meaning some occupants felt slightly warm. However, the average PMV throughout the Base Case was -0.695 (Figure 10), stating that the thermal sensation was slightly cool. The implentation of a chilled ceiling panel reduced

the PMV by 0.385 to -1.08, which was the lowest PMV value out of all cases. Consequently the occupants would feel a thermal sensation of 'Cool'. This would thus provide better working conditions for the occupants. Case 3 also had a similar thermal sensation of -0.94; again 'Cool'.
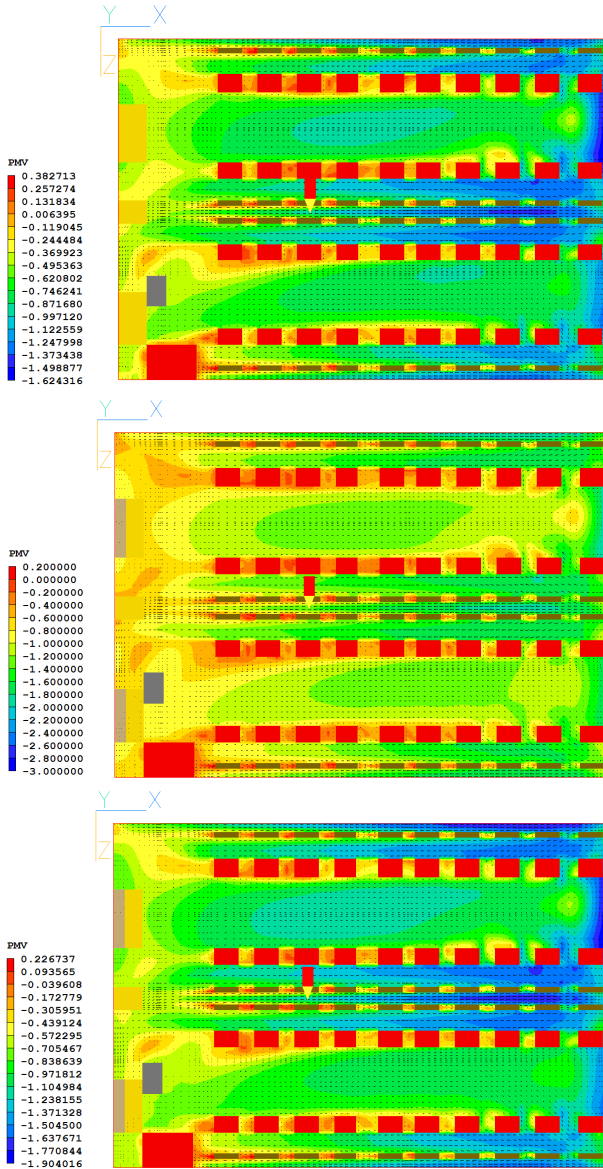

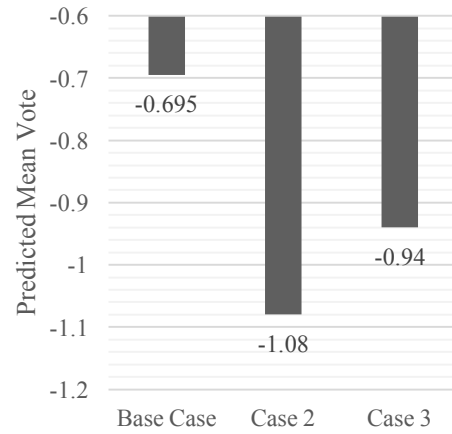Figure 9: Temperature profiles at E-E'; Base Case (top), Case 2 (middle), Case 3 (bottom).

Figure 10: Predicted Mean Vote average at slices E-E'; for base case, case 2 and 3.

Conclusions

Based on the literature reviewed, a clear assumption was made that a building with the application of thermal mass could reduce the interior temperature and consequently reduce the cooling load of the room. This could impact on the thermal sensation of the occupants. Therfore, this study aimed to examine whether the assumptions made from the mass of literature was verified, with the aid of the CFD simulation results.

Principal Findings

- In all cases the air was observed to behave in very similar ways. The main patterns of airflow were as follows: the air entered the domain through the HVAC's outlets, propagating along the domain towards the right-side wall. The air hit the right wall and flowed down towards the floor, distributing equally either inwards towards the middle two rows of computers or outwards towards the outer two rows of computers. The internal heat gains heated to the air causing an increased buoyancy. Some air recirculated back towards the right-side wall, however much of the air was then extracted from the domain via the HVAC's intakes.

- The use of thermal mass in the ceiling and walls of the IT suite, resulted in a dramatic reduction in the interior temperature, of 4.91°C (17%) from the Base Case.

- The application of a chilled ceiling panel to the IT suite, resulted in a substantial decrease in temperature of 2.65°C (9.2%) from the Base Case.

- The overall average temperature for the Base Case, Case 2 and Case 3 were; 26.61°C, 24.07°C and 23.68°C, respectively.

- The addition of thermal mass and a chilled ceiling panel to the space, effectively decreased

the PMV value by 0.245 (26%) and 0.385 (35.6%), respectively, from the Base Case.

Limitations of This Study

This study focused on an IT suite at Loughborough University and attempted to mimic realistic conditions as far as possible. The simulations were based on several assumptions and parameters that were constant for each of the cases investigated. The occupancy levels of the IT suite were kept constant throughout all cases.

The assumptions made included the IT suite at full occupant capacity for all cases, including the distribution of occupants in the spaces throughout the day, representing a full computer lab session during semester periods. Thus partial occupancy was not studied. Furthermore, the profiles of the internal gains, such as the computers, lighting and equipment, were created to comply with the occupancy levels and corresponding activities.

Recommendations for Further Work

Further simulation work is needed to model the occupancy at different levels, to determine the relevant changes in air temperature, velocity and thermal comfort. Additional IT suite optimisation and design needs to be modelled to look at the implications for improvements to thermal comfort.

Subsequent work could be conducted over a long period, possibly months or years, to depict the behaviour of the occupants throughout a semester or working year, supported by capital or running costs to predict cost savings.

Acknowledgements

References

Antonopoulos, K.A., and Koronaki, E.P., 2000. Thermal parameter components of building envelope. *Applied Thermal Engineering 20(13),* 1193– 1211.

Balaras, C.A., 1996. The role of thermal mass on the cooling loads of buildings. An overview of computational methods. *Energy and Buildings 24(1),* 1-10.

CHAM. (2015) Technical Report: TR 001. [online]. Available from: <http://www.cham.co.uk/phoenics/d_polis/d_docs/tr001/tr001.htm#what>.

CHAM (2016) PHOENICS Encyclopedia. [online]. Available from: <http://www.cham.co.uk/phoenics/d_polis/d_docs/tr326/obj-type.htm#Blockage>.

CHAM Ltd. (2008a) PHOENICS Encyclopedia. [online]. Available from: <http://www.cham.co.uk/phoenics/d_polis/d_enc/encindex.htm>.

CHAM Ltd. (2008b) PHOENICS Encyclopedia. [online]. Available from: < http://www.cham.co.uk/phoenics/d_polis/d_docs/tr006/chap5.htm#chap5.2>.

Chen, Q., 1995. Comparison of Different k-e Models for Indoor Airflow Computations. *Numerical Heat Transfer 28(B),* 353-369.

CIBSE, 2007. Guide A: Environmental Design. Issue 2, London: Chartered Institution of Building Services Engineers.

Cook, N.J., 1990. The Designers guide to wind loading of building structures, Part 2: Static Structures. London: Butterworths.

Cook, M. and Lomas, K.J., 1998. Buoyancy-driven Displacement Ventilation Flows: Evaluation of Two Eddy Viscosity Turbulence Models for Prediction. *CIBSE A: Building Services Engineering Research and Technology 19 (1),* 15-21.

Durrani, F., Cook, M.J., Mcguirk, J.J. and Kaye, N.B., 2011. *CFD Modelling of Plume Interaction in Natural Ventilation.* In: Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney, 14-16 November.

Gebremedhin, K.G., and Wu, B.X., 2003. Characterization of flow field in a ventilated space and simulation of heat exchange between cows and their environment. *Journal of Thermal Biology 28,* 301-319.

Hussain, S., Oosthuizen, P.H. and Kalender, A., 2012. Evaluation of Various Turbulence Models for the Prediction of the Airflow and Temperature Distributions in Atria. *Energy and Building 48,* 18-28.

Jiang, Z., and Chen, Q., 2009. *Air Distribution Effectiveness with Stratified Air Distribution Systems.* ASHRAE Research Project (RP-1373). August revision of code article.

Launder, B.E. and Spalding, D.B., 1974. The numerical computation of turbulent flows. *Computer Methods in Applied Mechanics and Engineering 3,* 269-289.

Menter, F.R., 1992. Improved two-equation $k - \omega$ turbulence models for aerodynamic flows. NASA Technical Memorandum 103975. Available from: < https://ntrs.nasa.gov/archive/nasa/casi.ntrs.nasa.gov/ 19930013620.pdf>. Accessed on 25/11/16.

Nicol, F. and Wilson, M. 2010. *An Overview of The European Standard EN 15251*. In: Adapting to change: New Thinking on Comfort, Cumberland Lodge, Windsor, UK, 9-11 April 2010. London: Network for Comfort and Energy Use in Buildings.

Nielsen, P., Allard, F., Awbi, H., Davidson, L. and SCHALIN, A., 2007. *Computational Fluid Dynamics in Ventilation Design*, REHVA Guidebook No.10.

Niu, J.L., Tang, Y.M. and Mak, C.M., 2005. The application of computational fluid dynamics to the assessment of green features in buildings: Part 2: Communal sky Gardens. *Architectural Science Review 48(4),* 337 – 344.

Norton, T., Sun, D.W., Grant, J., Fallon, R. and Dodd, V., 2007. Applications of computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural industry: a review. *Bioresource Technology 98*, 2386 – 2414.

Yakhot, V. and Orszag, S.A., 1986. Renormalization group analysis of turbulence. In: International Congress of Mathematicians Proceedings, Berkeley, California, USA, 1986. Available from: <mathunion.org/ICM/ICM1986.2/Main/icm1986.2. 1395.1399.ocr.pdf>. Accessed on 24/11/16.

Yao, W.L., Fan, Q.Y., Zhang, Y.W. and Xue, Y.B., 2012. The Research of CFD applications for thermal fluid simulation of Building Environment. IBPSA Proceedings 2012. Available from: <www.ibpsa.org/proceedings/asim2012/0059.pdf>. Accessed on 23/11/16.