A TRNSYS-FLUENT COUPLED SIMULATION OF THE THERMAL ENVIRONMENT OF AN AIRPORT TERMINAL SPACE WITH A MIXING AND DISPLACEMENT AIR CONDITIONING SYSTEM

[1]Lesh Gowreesunker, [2]Savvas Tassou
Dept. of Mechanical Engineering, Brunel University, Uxbridge, UK
[1] Email: Baboo.Gowreesunker@brunel.ac.uk
[2]Email: Savvas.Tassou@brunel.ac.uk

ABSTRACT

This paper reports on the simulation of the indoor thermal environment and energy demand of an Airport terminal space in the UK. It compares the performance of a mixed conditioning system (MC) and a displacement conditioning system (DC) in the airport terminal, under two different control strategies, through a coupled TRNSYS-FLUENT quasi-dynamic simulation. TRNSYS simulates the AC unit and control systems, while FLUENT simulates the airflow inside the terminal space.

The results show that the heating performance of both systems is comparable, while DC is more efficient for cooling purposes. Overall, yearly energy savings in the range of 2-30% are possible, depending on the AC system configuration.

INTRODUCTION

In indoor environments with large open spaces and high ceilings, such as airports, the energy consumption of air-conditioning (AC) systems is of crucial importance. Conventionally, long-throw nozzles have been used to mix the thermal environment and provide a uniform distribution of the AC air. In these circumstances, the AC air is supplied at relatively high levels and high velocities, generating mixing in the space, such as in Terminal 1 of Chengdu Shuangliu International Airport or Barcelona International Airport. However, this method has been found to involve considerable energy wastage, as it unnecessarily conditions large volumes that do not require conditioning (Goodfellow *et al.*, 2001).

Hence, in order to reduce the drawbacks of long-throw nozzles in mixed conditioning systems (MC), displacement diffusers have been increasingly employed for the conditioning of large spaces, as it reduces the conditioned area (Zhang *et al.*, 2009). In displacement conditioning (DC), for cooling purposes, cold air is blown into the space at low velocity and low heights. Upon reaching a heat source, the warm air plume rises, displacing warm air to higher regions, where it is removed. For heating purposes, DC systems behave similar to a mixed conditioning system, due to the buoyancy effects of warm air. Heathrow Terminal 5 and the New Bangkok International Airport employ the displacement conditioning method.

In such situations, the International Energy Agency (IEA, 1998) recommends that the airflow inside the space is very important in the efficient calculation of the energy demand of the building, and therefore suggests the use of Computational Fluid Dynamics (CFD) for the performance evaluations of AC systems. Various commercial simulation packages (such as IES-VE® or Design builder®) have, on that basis, introduced CFD components to their already existing zonal interfaces. The CFD capability of such packages are however simpler that actual CFD solvers, as the grids are uniform hexahedral, simple turbulence models are used, and the coupling between the building and the control system is done only at the end of the simulations.

As a result, the authors of this paper developed a coupling code to link the Energy Simulation tool TRNSYS and the CFD package FLUENT. TRNSYS is a transient, modular simulation program that allows the evaluation of various energy systems, including HVAC analysis. Each component of the simulation is linked such that the outputs of one component are fed to the inputs of another component. The relations between the input and output of each component can be determined analytically, numerically or empirically. The successive substitution method is employed for the iterative procedure, and convergence is obtained when the outputs of each component vary within the tolerance limits set in the kernel (Arias, 2006).

FLUENT is a general CFD simulation tool which solves the governing Navier-Stokes equations to provide different solution fields for a particular domain. FLUENT offers a more detailed and flexible approach, compared to commercial building simulation CFD tools. The level of precision depends on the model discretisation and the specific models employed, and the more refined the simulation, the longer the simulation times. Thus, although the level of detail in using FLUENT is very advantageous, the simulation times must also be kept at manageable levels in order to obtain an efficient coupled simulation tool.

COUPLING OF TRNSYS AND FLUENT

As mentioned in the previous section, TRNSYS contains the necessary HVAC components required to adequately design an AC system, while FLUENT can only simulate the airflow. In other words, the benefits provided by one simulation tool are missing in the other software, such that an optimum building simulation model would be a complement/coupling of both these tools. Zhai *et al.* (2005) classified such coupling strategies as either static or dynamic.

A static coupling involves a ‘one-step’ or ‘two-step’ data exchange between CFD and the ES simulation tools, and is employed where the solution is not very sensitive to the data being exchanged. Conversely, a dynamic coupling consists of a continuous exchange of data during the simulation. The latter category can be further divided into a fully-dynamic coupling involving data exchange at each iteration, or quasi-dynamic coupling involving data exchange at the end of each time-step. A fully dynamic coupling is more time consuming than the quasi-dynamic coupling, due to the higher number of coupled iterations. In this study, a quasi-dynamic coupling strategy is employed, as described in Figure 1.

This coupling occurs via a script and results file. A script file (*.in) is generated by TRNSYS which contains all the journal information for FLUENT. It consists of a text file containing the CFD case files, the boundary conditions at each time-step and the outputs needed by the CFD solver. A results file (*.txt) is text file, generated by FLUENT. It contains the results initially set by the *.in file. The fact that both simulation tools can read and write text files, makes this coupling possible. Furthermore, because of the respective architectures of the softwares, it is simpler to allow TRNSYS to control the entire coupled simulation. This is accomplished by creating a custom-built FORTRAN component in TRNSYS that externally calls FLUENT at the end of each time-step.

The inputs from TRNSYS at time t^{th} are passed to FLUENT in the *.in file, where the latter undergoes the necessary CFD calculations, until convergence is reached within the CFD solver. The ‘inputs’ comprise of the external weather data (T_a & Solar insolation), heat gains and ventilation supply air temperature, used as boundary conditions in the airport. The ‘CFD outputs’ from FLUENT at t^{th} are then passed to the control system which influences the AC system in order to send the appropriate, ‘updated ventilation temperature’ at $(t+1)^{th}$ to complete the information loop. The ‘CFD outputs’ are the feedback temperature (T_f) for the control system and the appropriate temperature for the HVAC system return air (T_r) - refer to Figure 2.

AC Unit and Control System

The AC unit employed is a constant air volume system, and a PID controller is used as the control system. The AC unit consists of a cooling and heating coil with fresh air flow rate of 0.2 kg/s in all cases and total air flow rates of 0.5 kg/s and 0.8 kg/s for the DC and MC cases, respectively (CIBSE A, 2006). Thus, the mixing ratio between the ambient air T_a and the return air T_r is constant throughout the simulations. The integral, derivative and gain constants of the PID are 0.1hr, 0.1hr and 10% respectively. These parameters are maintained for both the DC and MC cases, and for all seasons.

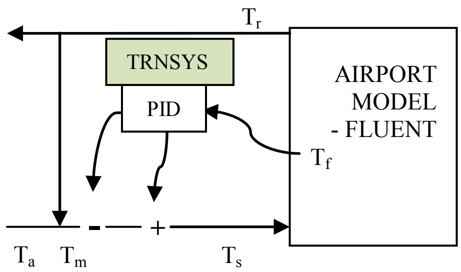


Figure 2. Schematic of AC unit

The comfort temperature range is 18-23 °C (CIBSE A, 2006), and the PID set point temperature is 21°C (optimum comfort temperature). The PID operates either heating or cooling, thus affecting T_s, based on the feedback temperature T_f, as follows:

- For T_f > 23°C; the PID calculates a lower supply temperature T_s to satisfy the cooling load.
- For T_f < 18°C; the PID calculates a higher

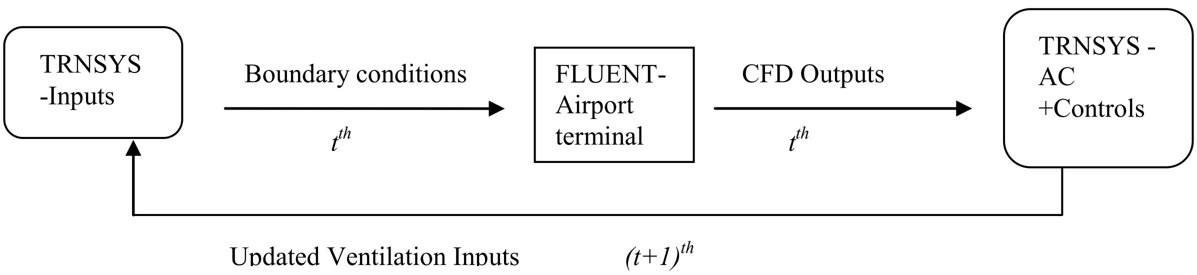


Figure 1. Information loop in TRNSYS-FLUENT coupling

supply temperature T_s to satisfy the heating load.

- During the comfort range (i.e. 18°C ≤ T_f ≤ 23°C), two different control strategies are investigated:

1. The PID produces T_s at 21°C, the optimum comfort temperature – referred to as '21-com'.
2. The PID is non-operational and the mixed air temperature T_m is supplied as T_s, i.e. a free-floating condition is imposed – referred to as 'free-com'.

Note that only T_s is varied by the PID controller because the AC unit is a constant air volume system.

SIMULATION MODELS

The airport terminal considered in this study is similar in geometry to London Heathrow Terminal 5. It contains a glazed surface on one extremity, a partially glazed roof and a large open space (see Figure 4). In reality, Heathrow Terminal 5 is served by a range of displacement diffusers placed on the floor, however, for the comparative purpose of this study, an assumed mixed ventilation system will also be investigated.

Radiation Model

As the airport terminal contains large glazed areas, an appropriate radiation model is required in the simulation. This study employs the gray Discrete Ordinates (DO) radiation model to model the radiation exchange within the building envelope. This method solves the radiative heat transfer equation for a finite number of discrete solid angles, and can consider absorption and scattering effects of the participating medium.

Solar radiation is the dominant radiation effect in a glazed building, and as proposed by the IEA Task 12 report, a suitable method to represent solar radiation in a space is through the use of heat fluxes in the building envelope in CFD (*Bryn and Schiefloe, 2006*). This is done in order to ensure the appropriate long-wave heat transfer modelling at night, which otherwise would require the non-gray modelling of radiation, increasing the complexity and simulation times of the model.

As a result, the solar radiation intensity on the floor and in the glazed surfaces are given by Equations (1) and (2), assuming Kirchhoff's law of radiation.

$$A_{floor}\,\dot{q}_{floor} = \alpha_{floor}\,G\left[\left(\tau_{glaze}\,A_{glaze}\right) + \left(\tau_{roof}\,A_{roof}\right)\right] \quad (1)$$

$$\dot{q}_{glaze} = \varepsilon\,\sigma\left({T_{sky}}^4 - {T_{ext}}^4\right) \quad (2)$$

The ambient/solar conditions are obtained from the Meteonorm weather data files C-37790 for London, and the sky temperatures are calculated from TRNSYS Type 69b.

CFD Model

The airport geometry is modelled as 2D (refer to Figure 4), where the roof, floor and glazed surfaces are also meshed in order to account for their thermal mass. A 2D geometry is chosen on the basis of the constant cross sectional area of the Terminal building and in order to minimise the computational time. The external surface of the floor is adiabatic, while the roof and glazed external surfaces are bounded by convective heat transfer conditions. The heat transfer coefficient is 25W/m^{2}K (CIBSE A, 2006). The ventilation inlets are defined as 'mass flow inlets' and the air-return is defined as a 'mass-flow-outlet', both with the same mass flow rates in order to conserve mass. The inlet size of the DC diffuser is 2m high and the return size is 4m high, while the MC inlet size is 0.36m and return size is 0.45m each. Each square (0.25m^2 and 1m high) mimics the occupants of the building and are bounded by heat fluxes, defined by Figure 3, adapted from the works of Parker *et al*. (2011).

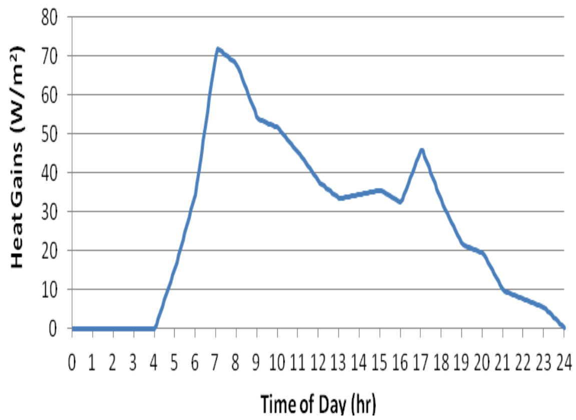


Figure 3. Internal occupancy heat gain

Figure 3 shows that the internal heat gains are highly non linear due to the non-uniformity in the occupants' schedule of the Terminal's in a day. There are peak heat gains at 07:00 and 17:00, and the airport is closed between 24:00 and 04:00. The AC unit is therefore not operational during the non-occupied hours during the night. In this study, the heat gain/ occupancy schedule is maintained for all seasons.

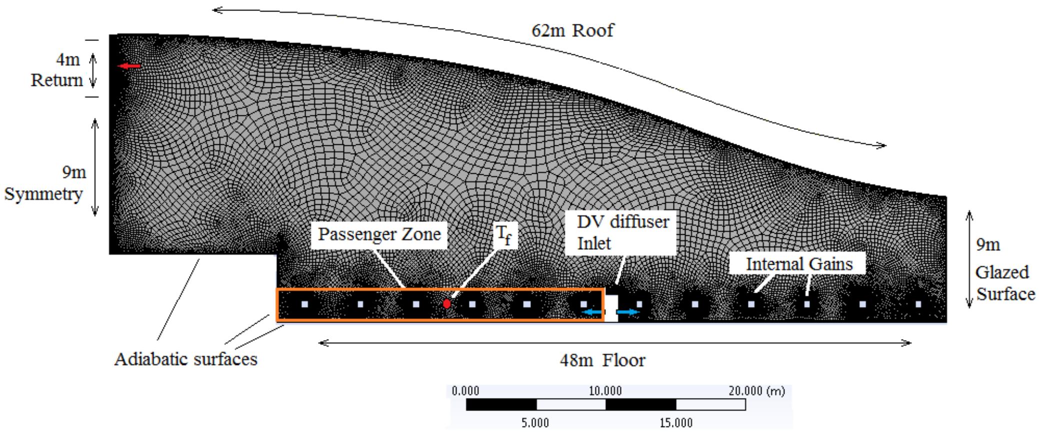


Figure 4. (a) Displacement Conditioning (DC) CFD grid

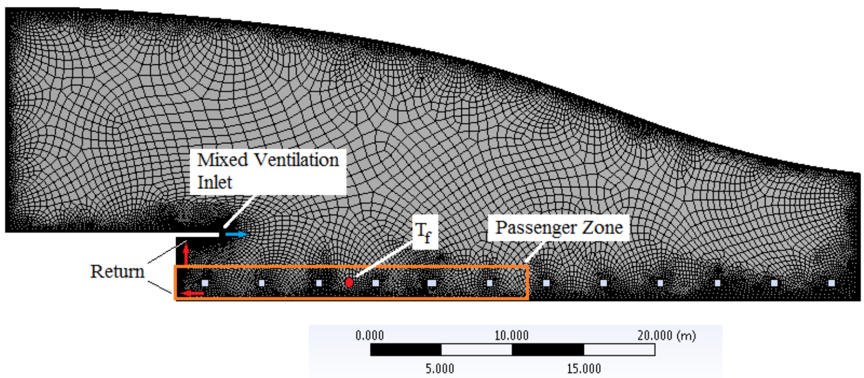


Figure 4. (b) Mixed Conditioning (MC) CFD grid

As mentioned previously, the CFD component in the coupled simulation is computationally expensive, and therefore impractical to employ very small time-steps in the simulation. As a result, the meshes for both the MC and DC cases, were designed based on the L_2 norm principle. The L_2 norm is used to quantify the errors in temperatures and velocities as the mesh is gradually relaxed from a benchmark model (Gowreesunker & Tassou, 2013). The benchmark models were obtained from grids which produced a y^+ values of 3.2 & 4.0 and a global Courant numbers of 1.55 & 2.05, for the DC and MC cases respectively.

Using the L_2 norm process to test for the spatial and temporal convergence, a time-step of 360s was obtained together with the meshes shown in Figures 4(a) and (b), which resulted in a temperature error < 0.5°C and velocity error < 0.15m/s. The meshes were designed using the ANSYS Workbench Design Modeller® automatic meshing algorithm, and are of the order of 5cm near the surfaces and 70cm in the bulk of the air domain. The RNG k-ε turbulence model, the SIMPLE algorithm and the body-force weighted discretisation schemes are used in the CFD models. The passenger zone represents the critical area where the indoor environment control is most important (Liu *et al.* 2009), and thus the location of the feedback temperature T_f. It should be noted that the time-step in TRNSYS is also 360s, to ensure the proper synchronisation between TRNSYS and FLUENT.

The building material properties are given in Table 1 (Note: Kirchhoff's law is applied to the radiation properties, i.e. ε = α).

Table 1.
Material properties

	Floor	**Glazing**	**Roof**
Thickness (mm)	13	30	30
ρ (kg/m^3)	1700	140	140
λ (W/mK)	0.8	0.03	0.03
c_p (J/kgK)	850	840	840
ε_{ext}	-	0.16	0.16
ε_{int}	0.5	0.2	0.2
τ	-	0.5	0.01

SIMULATION RESULTS

The DC and MC cases were run for 100 hours under distinct weather conditions, representing the summer, winter and an intermediate seasons, under the two control strategies '21-com' and 'free-com' . Due to the large computing times involved, the concept of Heating Degree Days (HDD) and Cooling Degree Days (CDD) were then used to extrapolate for the annual energy demands. The computing time for each season was approximately 14 hours.

Results – DC Case

Figures 5 show the generic temperature contours inside the terminal using a displacement diffuser, during the cooling and heating modes, respectively.

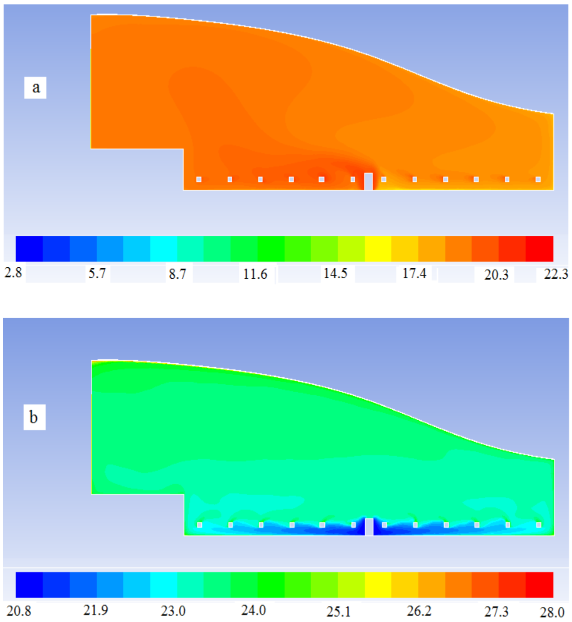


Figure 5. (a) DC Heating Mode,
(b) DC Cooling Mode

During heating, the DC system supplies warmer air to the space and because of the low supply velocity, the buoyancy effects dominate. As a result, the warm air rises and mixes within the entire air domain. During cooling, the low velocity and colder conditions of the supply air spreads on the floor, causing a separation of the conditioned zone from the rest of the unoccupied space. The stratification layers of the indoor air can be clearly observed, with a maximum temperature difference of ≈ 5.5°C between the floor and the roof. This shows the air-control effectiveness of using displacement diffusers during cooling, where the cooling potential of the AC can be specifically directed to the conditioned zone.

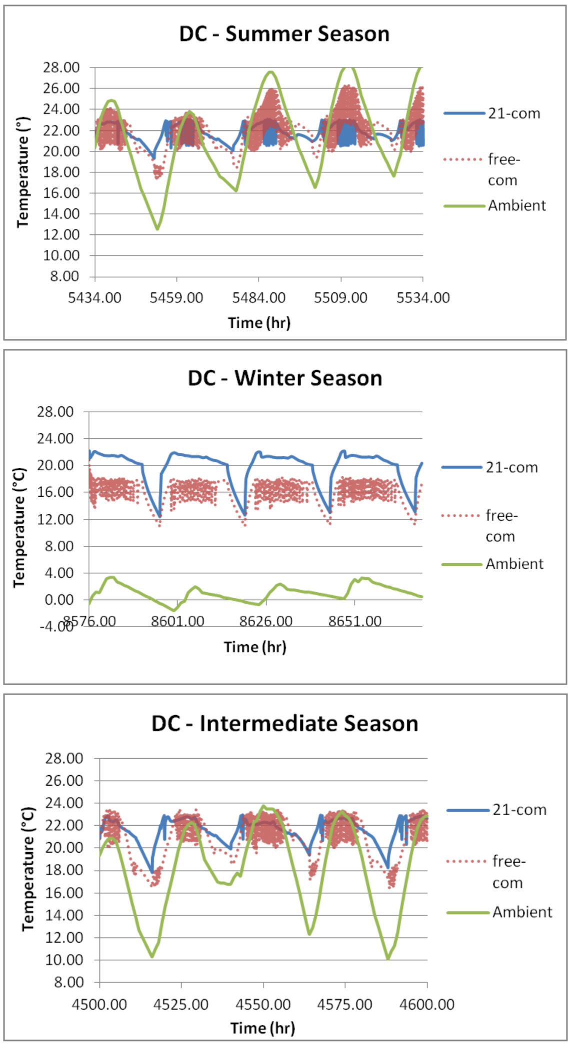


Figure 6. Zone (T_f) and ambient temperatures over the seasons – DC Case

Figures 6 show that using a temperature of 21°C within the comfort range produces a more comfortable and uniform thermal indoor environment over the three investigated periods. Using the free-floating condition, the AC unit is able to provide a comfortable environment only during the intermediate period, while the building overheats by about 3°C during the summer and undercools by about 3°C in the winter.

Results – MC Case

Figures 7 show the generic temperature contours using the long-throw nozzle inside the terminal under the heating and cooling modes.

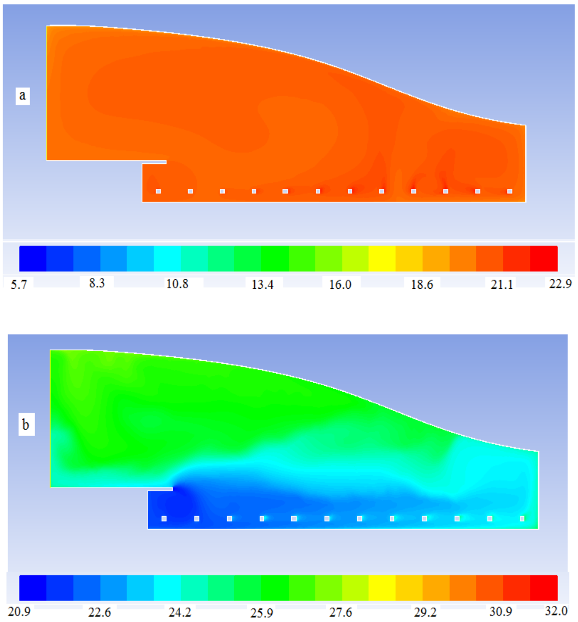
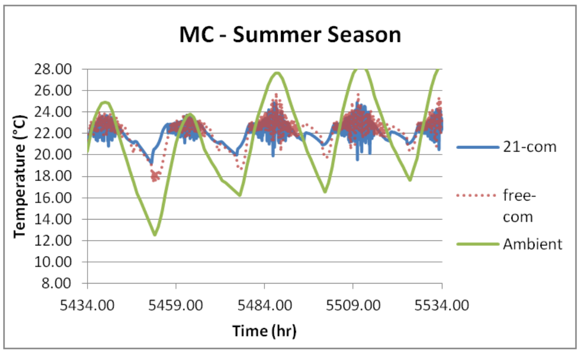
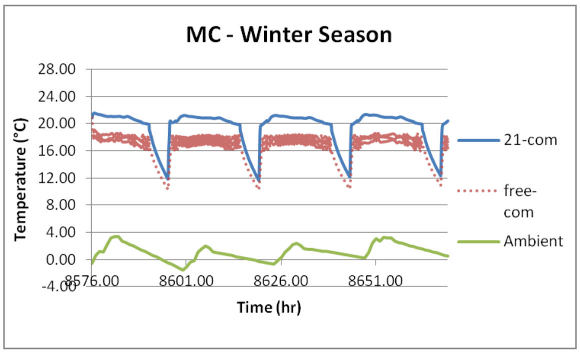


Figure 7. (a) MC Heating mode,
(b) MC Cooling mode

In the heating mode, the indoor space is fully mixed and the underlying mixing airflow path can be seen by closely observing Figure 7(a). During cooling, the space is not fully mixed and two distinct temperature zones can be observed, partly due to the geometry of the space. However, in contrast to the DC system, the MC system cools a much larger volume of unoccupied space, leading to cooling inefficiencies.





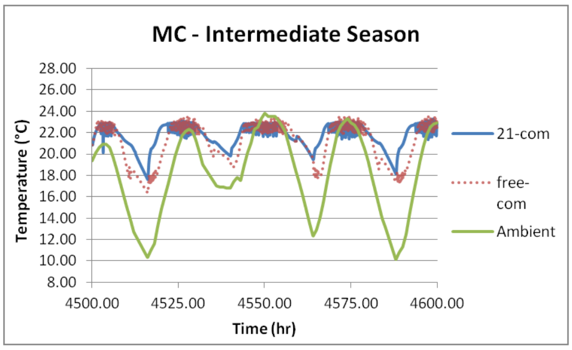


Figure 8. Zone (T_f) and ambient temperatures over the seasons – MC Case

Similar to the DC system, Figures 8 show that using a MC system with 21°C within the comfort range, provides a more comfortable and uniform indoor thermal environment. Employing the free-floating condition similarly attains the comfort criteria only during the intermediate season; undercools by 2°C during winter; and overheats by 3°C during summer.

It can generally be observed that the temperature trends for both systems are similar for the intermediate and winter periods. However, during summer, the DC system produces a relatively higher number of overheating hours (121hrs) compared to the MC system (96hrs) over the summer, with a reference temperature of 23°C.

Annual Energy Demands

The yearly energy demands are obtained from the concept of HDD and HDD, as proposed by *Day* (*2006*). The concept of degree days relies on the choice of a base temperature for heating and cooling, respectively. The base temperature for HDD refers to the ambient temperature below which heating is needed, whilst for CDD, it refers to the ambient temperature above which cooling is needed. Degree days include the summation of temperature differences over time, and therefore capture both the magnitude and duration of ambient temperatures.

Referring to Figures 6 and 8, the base temperature for the HDD and CDD, for the '21-com' scenario, is taken to be 15°C and 18°C, respectively. For the 'free-com' scenario, a temperature of 21°C is used as the base temperature for both the HDD and CDD.

Table 2.
HDD and CDD for the simulated periods and the entire year

	21-com		Free-com	
	HDD	CDD	HDD	CDD
Intermediate	2.49	6.27	12.89	1.70
Summer	0.29	13.79	5.75	7.02
Winter	46.35	0.00	66.10	0.00
Yearly	1859.12	120.10	3764.81	34.07

Degree days offer the benefits of simplicity and speed of use, and the ability to estimate building performance for different weather conditions. However, the concept only provides approximate results based on the assumption of average yearly conditions. It is used in this study because of the comparative nature of the results.

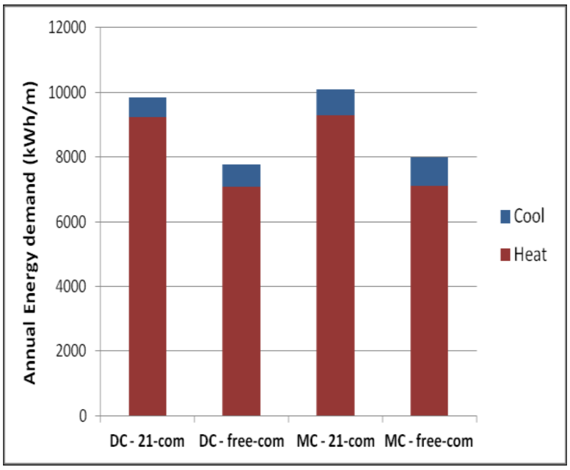


Figure 9. Annual energy demand with the DC and MC systems, and the '21-com' and 'free-com' strategies

Figure 9 shows that the cooling energy demand of all the configurations are relatively small compared to the heating demand of the airport terminal. This is mainly due to the UK weather conditions whereby the HDD are much larger than the CDD. Employing a DC-21-com configuration provides a reduction of 35% in cooling energy, while the DC-free-com saves 28% in cooling energy, with respect to the MC-21-com and the MC-free-com configurations, respectively. This therefore shows the cooling effectiveness of using the displacement diffusers within the airport terminal space.

The DC-21-com and MC-21-com, and MC-free-com and MC-free-com, each, consume very similar heating energies. This is due to the air mixing propensity of both systems in heating mode, where the two systems are effectively behaving as the same system.

It can be observed from Figure 9 that the DC-free-com consumes the least annual amount of energy. Thus, with respect to the DC-free-com configuration (used as benchmark), the DC-21-com, MC-21-com and MC-free-com configurations, respectively, consumes 27%, 30%, and 3% more annual energy.

It is interesting to note due to the dominant and similar annual heating energy demand for both systems in the UK weather, there is not a significant difference between the annual energy demand of the MC and DC systems, under the same strategies

CONCLUSION

This paper numerically investigates the energy performance and indoor environment control of a displacement diffuser and a long-throw nozzle in an airport terminal space. A coupled TRNSYS-FLUENT model has been developed and used to determine the performance of the building under two different control configurations. TRNSYS is used to simulate the AC unit and the PID control system, while FLUENT is used to simulate the indoor air-flow and radiation.

In this study, because of the absence of experimental data, a full validation of the models is not performed. Nonetheless, the errors in the model are quantified in terms of the L_2 norm study, the inputs are obtained from CIBSE and ASHRAE guides, and the widely validated RNG k-ε turbulence model is employed in CFD. Furthermore, this study specifically depicts a comparative analysis of the performance of the AC systems, rather than an absolute value analysis.

The results show that a DC system is more effective in cooling purposes, while the heating potential is similar to a MC system. Employing a free-floating condition when the indoor space is within the comfort range is more energy efficient than using a supply temperature of 21°C. However, using the latter strategy provides a more comfortable environment. Therefore, a balance between the comfort level and the AC energy consumption is crucial when determining an optimum AC configuration for a building.

NOMENCLATURE

A = Area (m^2)
$\dot{q}$ = Heat flux (W/m^2)
α = Radiation absorptivity
G = Solar irradiation (W/m^2)
τ = Radiation transmissivity
ε = Emissivity
σ = Stefan-Boltzmann constant
T = Temperature (°C or K)
ρ = Density (kg/m^3)
λ = Thermal coductivity (W/mK)
c_p = Specific heat capacity (J/kgK)

Subscripts

a = Ambient
r = Return
f = Feedback
m = Mixed
s = Supply
ext = External surface

ACKNOWLEDGEMENT

This work was made possible through sponsorship from the UK Engineering and Physical Sciences Research Council (EPSRC), Grant No: EP/H004181/1.

REFERENCES

Arias D.A. 2006. Advances on the coupling between a commercial CFD package and a component-based simulation program, 2nd National IBPSA-USA conference, Cambridge, MA, USA.

Bryn I., Schiefloe P.A. 2006. Atrium models for the analysis of thermal comfort and energy use, A report of the International Energy Agency (IEA) Task 12 – Building Energy analysis and design tools for solar applications, Project A.3.

Chartered Institute of Building Services Engineering (CIBSE) Guide A, 2006.

Day T. 2006. Chartered Institute of Building Services Engineering (CIBSE) TM-41: Degree days - Theory and Applications.

Goodfellow H., Tahti E. 2001. Industrial ventilation design guidebook. 1st Edition, Academic Press.

Gowreesunker B.L., Tassou S.A. 2013. Effectiveness of CFD simulation for the performance prediction of phase change building boards in the thermal environment control of indoor spaces, Building and Environment 59: 612-625.

International Energy Agency (IEA) Energy Conservation in Buildings and Community systems. Heiselberg P., Murakami S., Roulet C.A. 1998. Ventilation of large spaces in building – Analysis and prediction techniques, Annex 26: Energy Efficient Ventilation of Large Enclosures.

Liu J., Yu N., Lei B., Rong X., Yang, L. Research on indoor environment for the terminal 1 of Chengdu Shuangliu International airport, Proceedings of the Eleventh International IBPSA Conference, Glasgow, Scotland, 27-30 July 2009.

Parker J., Cropper P., Shao L. 2011. Using building simulation to evaluate low carbon refurbishment options for airport buildings, Proceedings of Building Simulation 2011: 12th Conference of International Building Performance Simulation Association, Sydney, 14-16 November 2011, pp. 554-561.

Zhai Z.J., Chen Q.Y. 2005. Performance of coupled building energy and CFD simulations, Energy and Buildings 37: 333-344.

Zhang T., Lee K., Chen Q. 2009. A simplified approach to describe complex diffusers in displacement ventilation for CFD simulations, Indoor Air 19(3): 255-267.