STUDYING THERMAL PERFORMANCE OF SPLIT-TYPE AIR-CONDITIONERS AT BUILDING RE-ENTRANT VIA COMPUTER SIMULATION

T.T. Chow, Z. Lin, Q.W. Wang and J.W.Z. Lu
City University of Hong Kong
Tat Chee Avenue, Kowloon Tong, Hong Kong
bsttchow@cityu.edu.hk

ABSTRACT

The use of split-type air-conditioners in new apartment buildings becomes popular in Hong Kong. One requirement for their effective use is satisfactory heat rejection at the outdoor condensing units. When a group of outdoor units is working together, the heat released by one condenser may affect the heat rejection rates at the others, and hence may deteriorate their performance. At the layout-design stage, an HVAC engineer often finds it difficult to predict such a thermal effect, say, when over a hundred of these outdoor units are to be placed inside one single re-entrant of a high-rise building. This paper presents a simulation approach to tackle the problem. The computation technique is demonstrated by its application to evaluate the effect of proper spacing between adjacent condensing units.

INTRODUCTION

Most residents of Hong Kong live in high-rise apartment buildings. New apartment buildings in the city are typically comprised of a number of residence towers standing on a common podium garden (see Figure 1). Covered car-park, shopping mall, and other supporting facilities are often found below the podium level. The floor plan of each residence tower is based on a cruciform design - with its four wings radiating out from the building core. At the building core are the communal space like lift shafts, lift lobbies, corridors, meter rooms, and public stairs etc. Typically there are eight flats on a single floor, two at each wing. A narrow but deep re-entrant exists at each wing between the neighbouring flats [1]. These re-entrants provide ventilation and natural light to the kitchens and the toilets of the apartments.

In recent years, split-type air-conditioners are widely adopted as a standard provision in new apartments to serve the living/dining rooms and the bedrooms. The distinct advantages of split-type units lie in the quiet operation and the flexibility in multi-room services [2]. By placing the outdoor condensing units inside the re-entrants, the provision can retain a clean and tidy external look of the building – not being spoiled by the window-mounted air-conditioners as it was used to be in the previous design.

However, it is known that for split-type units, effective cooling performance of the indoor air-conditioning units requires effective heat dissipation at the outdoor condensing units. For a 30-storey apartment building, it is not surprising to have over a hundred of these outdoor units being placed inside a single re-entrant. Heat energy released altogether brings about a natural upward flow of air. Depending on the actual layout of the outdoor units, air temperature at the upper part of the re-entrant may reach an unacceptably high level. This could in turn downgrade the air-conditioner performance because of the unsatisfactory condenser-cooling condition, or even cause accidental trip of the air-conditioners.

ENERGY PERFORMNANCE OF SPLIT-TYPE AIR-CONDITIONERS

Performance of a split-type unit can be described by its coefficient of performance (COP). The value of COP varies with the instantaneous working conditions, for instance, the room air condition and the condenser-cooling condition. An air-conditioner performs better at higher room temperature (T_r), and lower condenser on-coil temperature (T_o). The COP of a split-type unit can be related to T_o by the following linear regression equations at different room temperature conditions:

At $T_r = 23^oC$,

$$COP_{23}(T_o) = 4.825 - 0.0687T_o \tag{1}$$

At $T_r = 25^oC$,

$$COP_{25}(T_o) = 5.153 - 0.0738T_o \tag{2}$$

Figure 1 Plan view of 11 residential towers stood on a garden podium

At $T_r = 27^oC$,

$$COP_{27}(T_o) = 5.241 - 0.0742T_o \tag{3}$$

These equations were deduced from manufacturer data [3]. It applies well in the range of T_o between 25 to 45^{o}C. A condenser on-coil temperature higher than 45^{o}C may lead to abnormal refrigerant working pressure and stop the air-conditioner from functioning. Hence for a given layout of the condensing units, if the on-coil temperatures of the individual units are known, the performance of all air-conditioners can be estimated.

CFD SIMULATION WITH k-ε MODEL

Computational Fluid Dynamic (CFD) analysis is an ideal tool to help understand the complex phenomena of flow problems. Among the various choices, the k-ε model of turbulence [4] is so far the one most popularly used. By solving numerically the Navier-Stokes, continuity and energy equations, this method offers the best trade-off between accuracy and computational effort. For solving a steady-flow thermal problem with the 2-equation k-ε model based on the Boussinesq approximation, the basic transport equations in a 3D vector space are [5]:

Continuity

$$\frac{\partial \bar{U}_i}{\partial X_i} = 0 \tag{4}$$

Momentum

$$\bar{U}_j \frac{\partial \bar{U}_i}{\partial X_j} = -\frac{1}{\rho}\frac{\partial \Pi}{\partial X_i} + \frac{\partial}{\partial X_j}\left[(\nu + \nu_t)\left(\frac{\partial \bar{U}_j}{\partial X_i} + \frac{\partial \bar{U}_i}{\partial X_j}\right)\right] - \beta g_i \bar{\theta} \tag{5}$$

Thermal Energy

$$\bar{U}_i \frac{\partial \bar{\theta}}{\partial X_i} = \frac{\partial}{\partial X_i}\left(\kappa + \frac{\nu_t}{\sigma_\theta}\right)\frac{\partial \bar{\theta}}{\partial X_i} + \bar{q}_\theta \tag{6}$$

Turbulence Kinetic Energy

$$\bar{U}_j \frac{\partial \bar{k}}{\partial X_j} = \frac{\partial}{\partial X_j}\left[\left(\nu + \frac{\nu_t}{\sigma_k}\right)\frac{\partial \bar{k}}{\partial X_j}\right] + \nu_t\left(\frac{\partial \bar{U}_j}{\partial X_i} + \frac{\partial \bar{U}_i}{\partial X_j}\right)\frac{\partial \bar{U}_i}{\partial X_j} + \beta g_i \frac{\nu_t}{\sigma_\theta}\frac{\partial \bar{\theta}}{\partial X_i} - \bar{\varepsilon} \tag{7}$$

Dissipation Rate of Turbulence Kinetic Energy

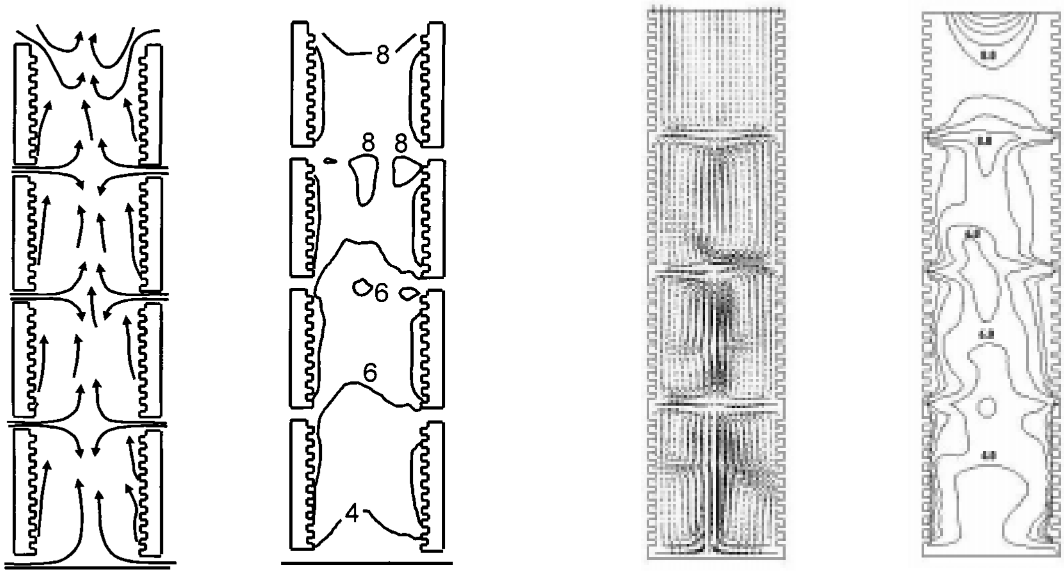
$$\bar{U}_j \frac{\partial \bar{\varepsilon}}{\partial X_j} = \frac{\partial}{\partial X_j}\left[\left(\nu + \frac{\nu_t}{\sigma_\varepsilon}\right)\frac{\partial \bar{\varepsilon}}{\partial X_j}\right] + \frac{\bar{\varepsilon}}{\bar{k}}\left[C_1\nu_t\left(\frac{\partial \bar{U}_j}{\partial X_i} + \frac{\partial \bar{U}_i}{\partial X_j}\right)\frac{\partial \bar{U}_i}{\partial X_j}\right] - C_2\frac{\bar{\varepsilon}^2}{\bar{k}} + C_3\frac{\bar{\varepsilon}}{\bar{k}}\beta g_i \frac{\nu_t}{\sigma_\theta}\frac{\partial \bar{\theta}}{\partial X_i} \tag{8}$$

Eddy Viscosity

$$\nu_t = C_D \frac{\bar{k}^2}{\bar{\varepsilon}} \tag{9}$$

Quality of the simulation results relies very much on the appropriate use of the simulation software, the model parameters, the numerical grid, the boundary conditions, as well as the underlying assumptions to the scale of the problem. It is therefore important to examine the appropriateness of the simulation approach before its extended applications in related practical studies. Published experimental data related to buoyancy induced airflow in and around semi-enclosed space of a tall building was therefore used to verify the correct application of the software CFX-4.2 [6] and the numerical scheme in our work. Attempts were made to use the CFX-4.2 program to simulate experimental conditions for studying discharges from gas water heaters into a light well of a high-rise apartment building. It was assumed that heat energy was removed only by convection through the openings at the enclosing walls. A 1:100 laboratory rig was used in the measurements, of which the details had been reported by Kotani et al. [7]. During the experiment, air temperature was recorded by thermocouples and air velocity by anemometer. Very good matching between the simulation and experimental results were found. Some comparisons of the airflow pattern and temperature rise pattern are shown in Figure 2. Numerical values of the empirical constants used in the simulation model were C_1=1.44, C_2=1.92, C_3=0.0, C_D=0.09, σ_k=1.0, σ_ε =1.3, and σ_θ=0.9.

Experimental results Simulation results



(a) (b) (c) (d)

Figure 2 Experimental validation, (a) and (c) shows airflow pattern; (b) and (d) shows air temperature contours

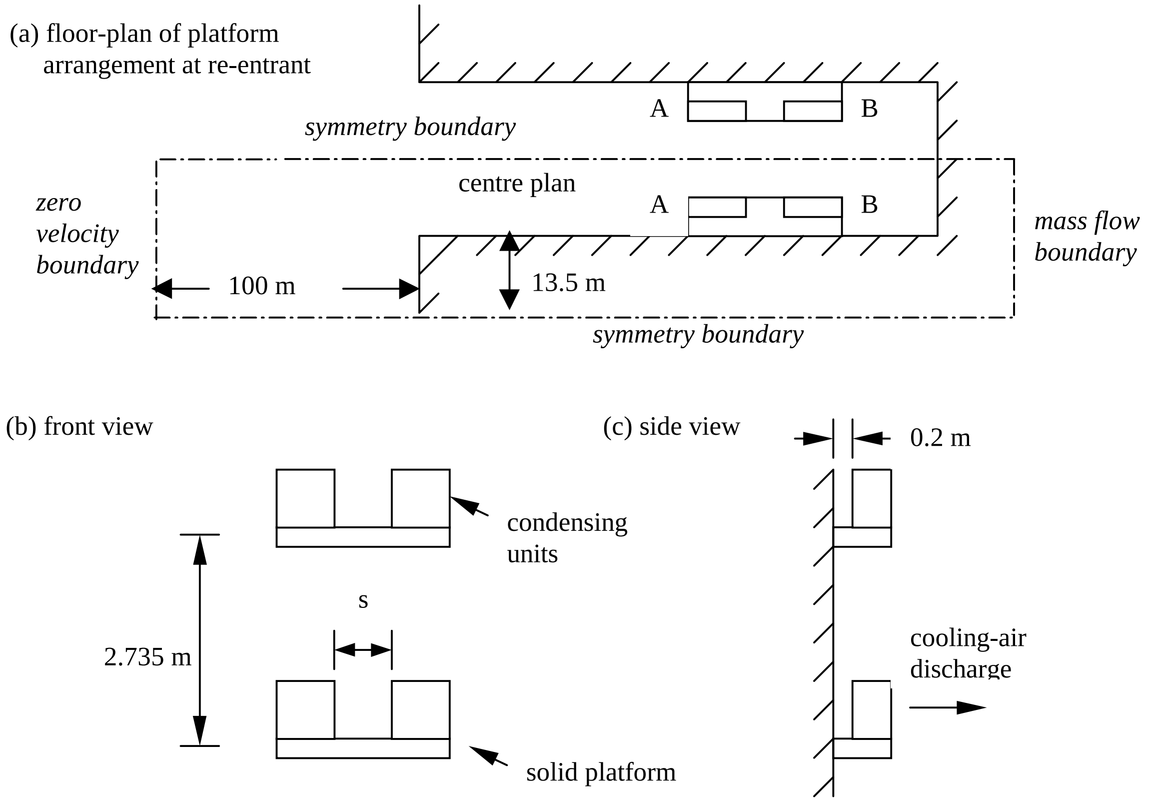


Figure 3 Arrangement of condensing units in building re-entrant (not to scale)

MODEL OF CONDENSER OPERATION

Figure 3 shows one possible arrangement of condensing units in a building re-entrant: 2 identical units being mounted on a common solid platform, which is then fixed onto a solid wall. A gap of 200 mm was left between a condensing unit and the wall surface. The re-entrant used in the study was one with 30 floor levels, and of dimensions 10 m by 3 m by 82.05 m. There were 4 condensing units per floor level. The two solid platforms were fixed onto the opposite walls. It is known that a saving of installation costs can be achieved by using smaller platforms, i.e. shorter distance "s" between the adjacent units. Yet to allow adequate heat dissipation, the performance of the air-conditioners was expected better for greater spacing between the two units. The sensitivity of this spacing was thus studied through a CFD analysis.

In practice, a prediction of the operating conditions of the condensing units can be complicated by the fact of diversity – the air-conditioners may not be all in operation at any moment. Also for those in operation, their thermal loads do vary. Simplifications and assumptions have to be made to develop a steady-state computational model for the study. Those adopted in our model include:

i) all identical air-conditioners are in operation and are under steady cooling load at 50% of their rated capacity;
ii) all wall surfaces of the re-entrant are flat and air-tight;
iii) the outdoor temperature remains steady at 33^{o}C, i.e. at the summer design condition of Hong Kong;
iv) there is no wind effect; the upward flow of air is merely induced by the buoyant effect;
v) the effect of solar radiation on the flow field is negligible.

In the numerical analysis, only one half of the re-entrant (up to the centre plan) was modelled using a 3D rectangular grid (occupying the enclosed space in Figure 3-a), taking the advantage of the symmetry in the re-entrant shape and the condensing unit arrangements. The "casing" of each outdoor unit was represented by 4 solid surfaces, leaving the front and rear ends "open" for cooling airflow. Cooling fans were modelled by adding body forces to the enclosed cells. Volumetric heat energy was added to the enclosed cells to simulate the condenser heat dissipation. To represent a no-wind condition, air velocity at a distance of 100 m away from the re-entrant front was taken as zero. This distance of the zero-velocity boundary was adequate since a change of the distance from 100 m to 5 m caused only less than 0.5°C deviation in the simulation results of on-coil temperature. Mass-flow boundaries were used at 40 m above the re-entrant top and at 5 m downstream the rear-wall of the re-entrant. Symmetry boundaries were used at the two sides, i.e. at the centre plan and at 13.5 m away from the front edge of the re-entrant. Logarithmic wall profile was adopted. A structured grid over 170,000 cells was finally used. Individual simulations were performed using CFX-4.2 with different condenser spacing.

Figure 4 shows the results of condenser on-coil temperature against floor level for the two condenser positions A and B and for 3 different cases of spacing at 0.1 m, 0.4 m and 1.2 m respectively. It can be observed that the on-coil temperature generally rises with floor level except at the top two levels, where the temperature condition was found influenced by the ingress of outside air over the top. Comparatively, the temperature conditions at Position A were better than at Position B. Obviously this was because Position A was closer to the re-entrant front and therefore received more outside air.

The following describes a comparison of the three cases using an energy evaluation model.

ENERGY EVALUATION

Within a re-entrant, the on-coil temperature T_o of all condensing units are either equal to or above the outside air temperature. To consider the performance of "n" numbers of these condensing units as a group, a Condenser Group Performance Indicator (CGPI) can be used which is expressed as:

$$CGPI_{T_r}(T_{ref}) = \frac{100}{n}\sum_{i=1}^{n}\left[1-\left(\frac{COP_{T_r}(T_o)}{COP_{T_r}(T_{ref})}\right)_i\right] \tag{10}$$

This term describes the average percentage drop in COP of all air-conditioners under consideration, with respect to a referenced outdoor temperature T_{ref} and a specified room temperature T_r. "n" in this case is 120 (4 condensing units per floor for 30 floor levels). T_{ref} can be arbitrarily assigned, say in this study, the summer design condition of Hong Kong was chosen. It can be shown by applying equation (2) and equation (10) that for T_r = 25 °C and T_{ref} = 33 °C, the corresponding CGPI value is given by

$$CGPI_{25}(33) = 2.72T_{om} - 89.76 \tag{11}$$

where T_{om} is the mean value of all on-coil temperatures.

.

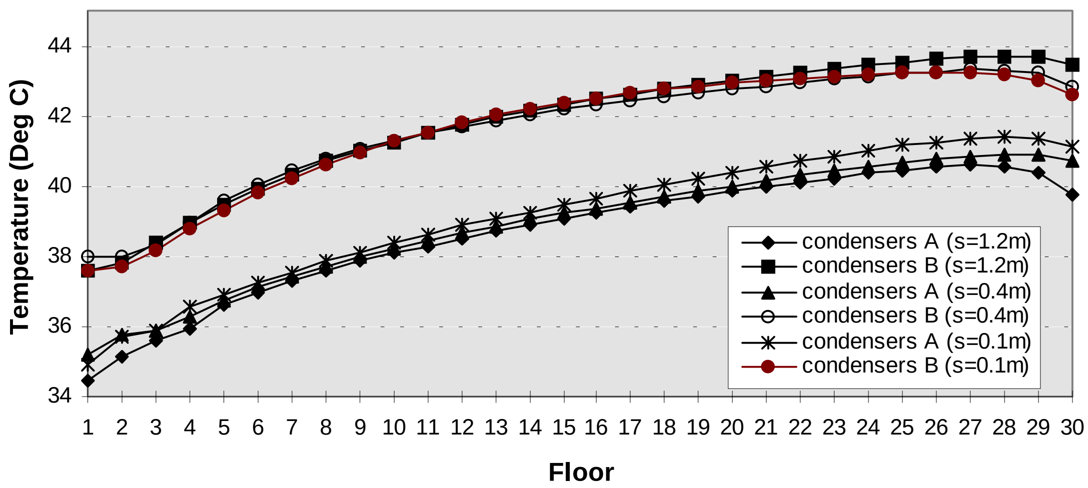


Figure 4 Simulation results of condenser on-coil temperature for different condenser spacing

Applying equation (11) to the three simulated cases, the CGPI values were found to be 19.62, 19.77 and 20.11 respectively for the spacing of 1.2 m, 0.4 m, and 0.1 m. Hence the average drop in COP was only further reduced by 0.76% if the spacing was increased from 0.4 m to 1.2 m (i.e. triple the distance). If the change of spacing was from 0.4 m to 0.1 m (i.e. shortening the distance to one quarter), the COP drop was further increased by 1.7%. The results therefore suggested that 0.4 m was a reasonable optimal spacing between the adjacent condensing units in this proposed arrangement.

Nevertheless, with this spacing, the overall average-drop in COP would be around 20%. This implies an overall increase of electricity consumption by around 25% compared with the referenced 33^{o}C on-coil temperature conditions. More desirable results could be obtained by separating the opposite platforms further apart, subjected to the maximum allowable refrigeration pipe-lengths inter-connecting the indoor and outdoor units. Similar numerical analyses could then be applied to alternative arrangements to sort out the best solution. For a given situation, an experienced user can complete these simulation analyses within a month. The methodology is therefore acceptable to the practical design process of a building project in terms of costs and time span.

This approach of evaluating group condenser performance, although is introduced to study the outdoor condensing units being placed in building re-entrants under the Hong Kong climatic conditions, the concept of CGPI actually can be extended to use in other places by selecting a relevant T_{ref} , or to use in evaluating alternative layout schemes of condensers when working in cluster.

CONCLUSIONS

CFD simulation is a very useful technique in the study of condenser heat dissipation within a building re-entrant. This paper introduced a simulation approach with the use of k-ε model together with an energy evaluation model in the analysis. The technique used in the numerical simulation was supported by experimental validation based on similar thermal studies. Practicability of the technique was demonstrated through an application example. The derivation of the Condenser Group Performance Indicator (CGPI) allows a representation of the overall performance, whereby multiple effects can be monitored by a single parameter. The required time span and expenses in this approach support its application to real building projects. HVAC engineers can use the mentioned technique to search for optimal design solutions.

ACKNOWLEDGEMENTS

This research project is financially supported by the Strategic Research Grant #7000667 from the City University of Hong Kong.

REFERENCES

[1] D. Cronin, "Investigation into the dynamics of waste air dispersal from high-rise residences", Proc. 30th ANZAScA Conf., Hong Kong, July 1996, pp.85-92.

[2] ASHRAE, ASHRAE Handbook: HVAC Systems and Equipment, Atlanta, 1996.

[3] T.T. Chow and C.K. Cheng, Survey report on performance of split-type air-conditioners with outdoor units placed at building re-entrants, Internal Report, Division of Building Science & Technology, City University of Hong Kong, 1998.

[4] B.E. Launder and D.B. Spalding, "The numerical computation of turbulent flow", Computer Methods in Applied Mechanics and Engineering, Vol.3, 1974, pp.269-289.

[5] J.H. Ferziger and M. Peric, Computational Methods for fluid dynamics, Springer, Germany, 1996.

[6] AEA, CFX 4.2 Flow Solver User Guide, Computational Fluid Dynamics Services, Harwell Laboratory, UK, 1998.

[7] H. Kotani, R. Satoh and T. Yamanaka, "Stack effect in light well of high rise apartment building", Proc. IIR Int. Sym. Air Conditioning in High Rise Buildings '97, Sep 1997, Shanghai, China, pp.628-633.

NOMENCLATURE

C_1, C_2	empirical constant in generation/ destruction term of ε-equation
C_3	empirical constant in buoyant term of ε-equation
C_D	empirical constant for eddy viscosity
CGPI	condenser group performance indicator
COP	coefficient of performance
g_i	gravitational constant in X_i-direction (m/s^2)
$\bar{k}$	mean turbulent kinetic energy (m^2/s^2)
n	number
$\bar{p}$	mean static pressure (N/m^2)

$\bar{q}_\theta$	mean volumetric heat source generation rate (kW/m^3)
T	temperature (^{o}C)
$\bar{U}_i$	mean velocity component in X_i-direction (m/s)
$\bar{U}_j$	mean velocity component in X_j-direction (m/s)
X_i , X_j	distance in cartesian co-ordinate (m)

Greek letters

β	volumetric expansion coefficient (K^{-1})
$\bar{\varepsilon}$	mean dissipation rate of $\bar{k}$ (m^2/s^3)
θ	temperature rise above ambient (K)
κ	thermal diffusivity (m^2/s)
ν	kinematic molecular viscosity (m^2/s)
ν_t	eddy viscosity (m^2/s)
Π	total pressure (N/m^2); $\Pi = \bar{p} + \frac{2\rho\bar{k}}{3}$
ρ	fluid density (kg/m^3)
σ_k	empirical constant of turbulent Prandtl number for k
σ_ε	empirical constant of turbulent Prandtl number for ε
σ_θ	empirical constant of turbulent Prandtl number for θ

Subscripts

m	mean
o	on-coil
r	room
ref	referenced
θ	in θ equation