CHEMICAL ENGINEERING TRANSACTIONS
VOL. 61, 2017

A publication of AIDIC
The Italian Association of Chemical Engineering
Online at www.aidic.it/cet

Guest Editors: Petar S Varbanov, Rongxin Su, Hon Loong Lam, Xia Liu, Jiří J Klemeš

ISBN 978-88-95608-51-8; **ISSN** 2283-9216
DOI: 10.3303/CET1761114

CFD Modelling Approach for the Optimum Cooling Scheme for the Partially Opened Corridors in a Commercial Centre

Guangyu Han*, Ye Gao

College of Aerospace and Civil Engineering, Harbin Engineering University, Harbin, China
983682075@qq.com

Thermal comfort at a partially opened corridor in a commercial center is studied by means of computational fluid dynamics (CFD) modelling. Cooling load is verified to improve the thermal comfort to a satisfied level. Results on the temperature profile and air flow distribution are compared for several options. Feasible cooling strategy is proposed based on the cooling performance, air distribution, energy saving, and operation impact.

1. Introduction

With the recent highlighting of global warming and the high amounts of pollutant emissions, the necessity for utilizing passive design strategies in buildings to achieve indoor thermal comfort is currently being given serious attention. This is highlighted in large modern commercial and office buildings, especially when they tend to benefit more from natural daylight, like corridors, which result in overheating and additional air conditioning loads. Corridors (as a feature of passive design strategies) in modern large buildings are incorporated into the built environment more and more frequently, in many different building types.

2. Background information

To provide a circulation space, rich daylight and other flexible indoor space, atria with surrounding corridors have been widely used in buildings, particularly large commercial buildings. But this feature design may introduce some environmental problems, such as excessive solar heat gain and high resultant temperature at top levels of occupant area close to the atrium. Such negative effect might be considered at early design stage so as to optimize the design. Computer modelling has widely been used as a prediction tool in building engineering and architectural design. However, there might be difficult to define the best cooling scheme at early stage for complex design. Computational fluid dynamics (CFD) has been widely used at all design stages, which may offer accuracy and cost effectiveness over empirical estimation. A commercial centre located in Guangzhou China has been opened for several years. But there are many complaints on the hot environment along many corridors surround the atriums designed for the commercial centre. There are opening gaps above the atriums and the corridors are connected with atriums with one meter high glass barrier, but the cooling performance is far behind the expectation with the current natural ventilation. To improve the thermal comfort, the alteration and addition (A&A) works to existing buildings is proposed. It is difficult to determine the cooling load for the partially opened corridors which is different to common enclosed rooms. Complete site test is impossible for the busy operation conditions. CFD study is proposed to support this A&A work for all the concerned corridors in this centre. The proposed CFD modelling approach will involve the cooling load verification and feasible cooling schemes with consideration of the general performance and impacts on the current operation.

3. Methodology for the CFD modelling

3.1 Domain

A typical corridor (25 m × 3 m × 3 m) is taken as a sample for the CFD modelling. The computational domain is 25 m × 3.6 m × 3.2 m with total cells of 799,008 (232 × 123 × 28).

Please cite this article as: Han G., Gao Y., 2017, Cfd modelling approach for the optimum cooling scheme for the partially opened corridors in a commercial centre, Chemical Engineering Transactions, 61, 697-702 DOI:10.3303/CET1761114

The cooling load should be first estimated and verified by the CFD simulation. Feasible A&A options (A, B and C) will be considered based on the corridor conditions. There are the following scenarios considered for the CFD study:

- Existing environmental simulation to verify the existing temperature environment
- Cooling load verification to achieve a satisfied thermal level
- Air supply from ceiling center along the corridor (option A)
- Air supply from ceiling side along the corridor (option B)
- Air supply from floor level along the corridor (option C)

CFD simulation results on the temperature, air flow, and PPD and PMW of the thermal comfort parameters for different options are used to assess the cooling performance.

For each cooling option, various glass barrier heights, velocity of air supply, shape and angle of the grille, distribution position are considered for different simulation scenarios. Moreover, construction difficulties and the impacts on the current operation are also ranking factors for final confirmation. In order to better explain the solution to the problem, Figure 1 serves as an example that is placed in section 'Boundary Conditions'.

3.2 Boundary conditions

The ambient temperature is 34.2 °C, and the cooling air is designed as 22 °C with total volume flow rate of 2.4 m^3/s to avoid condensation. According to the local weather data to choose a more conservative solar radiation heat load, direct solar radiation heat for 470 W/m^2 and diffuse radiant heat for 280 W/m^2 on one o'clock at noon. Total heat load for the typical is conservative taken as 75 × 24 W for occupants (0.4 m × 0.25 m × 1.8 m), 630 W for the equipment within the typical corridor, and 280 W/m^2 for solar radiation as in Figure. 1.

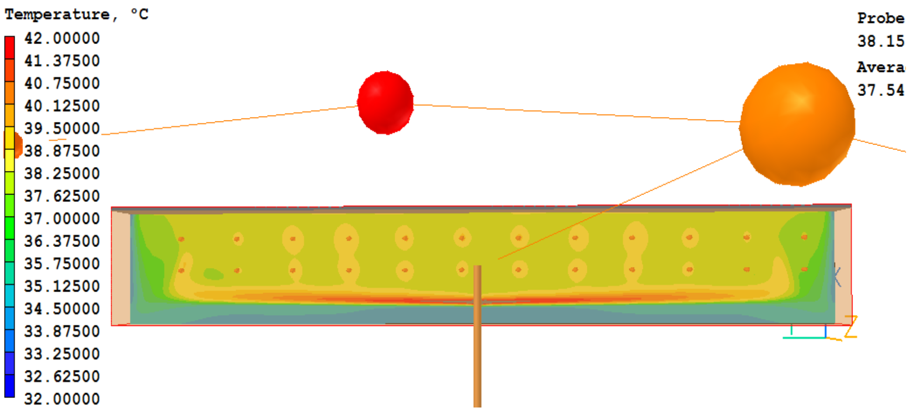

Figure. 1: CFD modelling

3.3 Turbulence model

The Reynolds numbers for most flows within the built environment are in the transition range and the airflow in a corridor is turbulent too. There is currently no universal turbulent model available that can reflect the behaviour of full range of complex turbulent flows observed in buildings. From the viewpoint of engineering use, simulation based on the Reynolds averaged equation is the most efficient and commonly used. The k-ε two – equation model is used to incorporate the effect of turbulence on the flow and they have been widely used in building ventilation and air-conditioning studies yielding many satisfactory results. The standard k-ε turbulence was applied with the HYBRID differencing scheme.

3.4 Radiation model

In a large enclosure, it is essential to consider surface heat exchange not only by natural convection, but also by long-wave thermal radiation. Both phenomena are governed by surface temperature. This is even true in an corridor, where the high surface temperatures on ceiling and other surface exposed to direct sun light are the major source of heat inside the corridor and contribute greatly to thermal stratification. Therefore, the long-wave redistribution of solar radiation in the corridor is vital to be considered in this CFD corridor modelling studies.

The radiative heat transfer within the representative corridor models was solved by radiative model called IMMERSOL. The absorption coefficient was set to be 1.0 and the 'Store Radiative energy fluxes' was switched ON. The emissivity of all the boundary surfaces was also set in addition to their surface temperatures. The internal walls which were of plastered bricks and the concrete floor elements were set to have emissivity of

0.9, whereas the emissivity of glazed walls and roof elements were to be 0.845.

3.5 Iteration and convergence

Apart from having sufficient numerical grid points and resolution, the accuracy of the solution obtained by CFD simulation also usually depends on the degree to which the converged solution satisfies the discretized equations. This can normally be assessed by the level of imbalanced error-sources within discretized equations.

A point in a flow domain at which the flow variables can be probed or monitored as the solution runs should be carefully set. In general, the probe is positioned at a location where flow is expected to have a high value of velocity gradient. Particularly in a large flow domain, a few trial runs should be carried out to identify the most critical location that would affect the overall solution of the flow field parameters in the domain. At this critical monitoring point, it is normally rather difficult to get converging solution at this spot and usually requires longer running time (or higher iteration number) for the values of the variables to become stable or constant.

Feasible options

To assess the cooling performance, there are many different parameters including the air temperature, air flow velocity, PPD and PMW of the derived thermal comfort parameters. But the most important key parameter is the temperature. Other parameters can be used to support the comparison.

Position of assessing point or area is another key factor to be considered. It is not reasonable to consider all the area for the whole corridor. The critical area is the most widely used area for passenger walking through the corridor.

Three feasible options as option A, option B and option C are compared with typical CFD results on the temperature along the corridor centre shown in Figure. 2, Figure. 3 and Figure. 4.

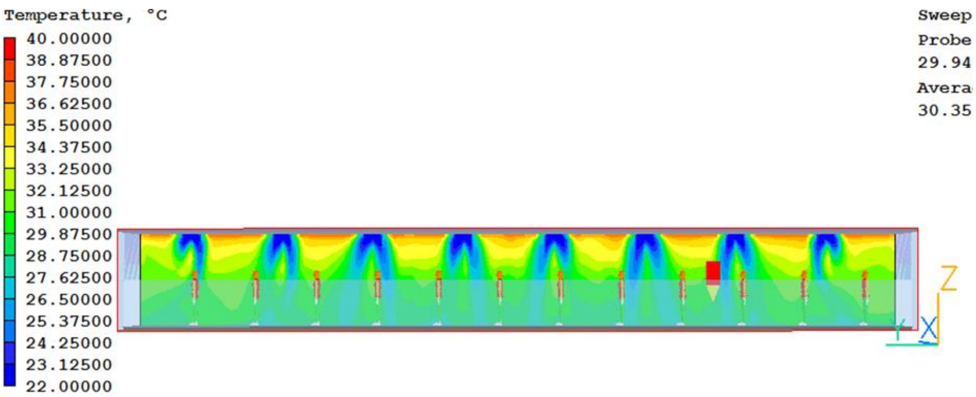

Figure 2: Temperature profile for air supply from ceiling – (option A)

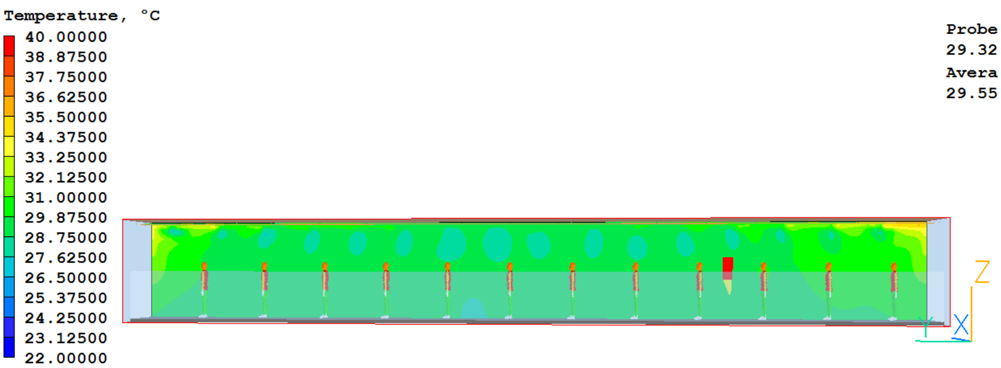

Figure 3: Temperature profile for air supply from ceiling - (option B)

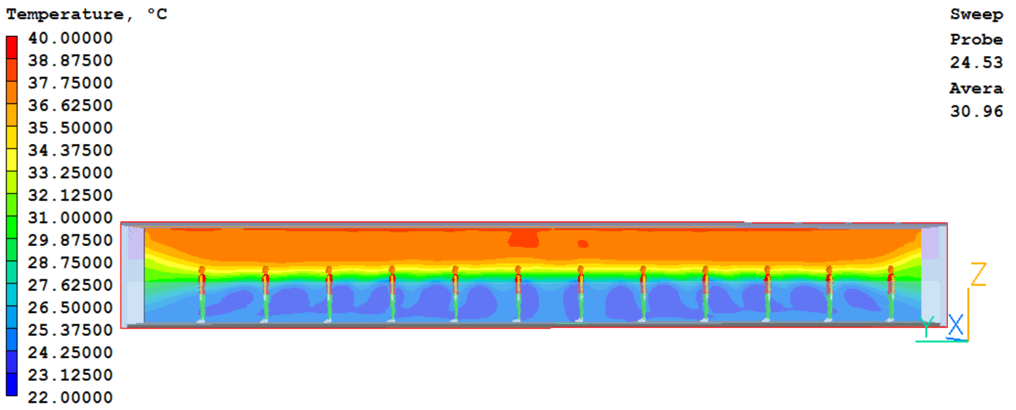

Figure 4: Temperature profile for air supply from floor level - (option C)

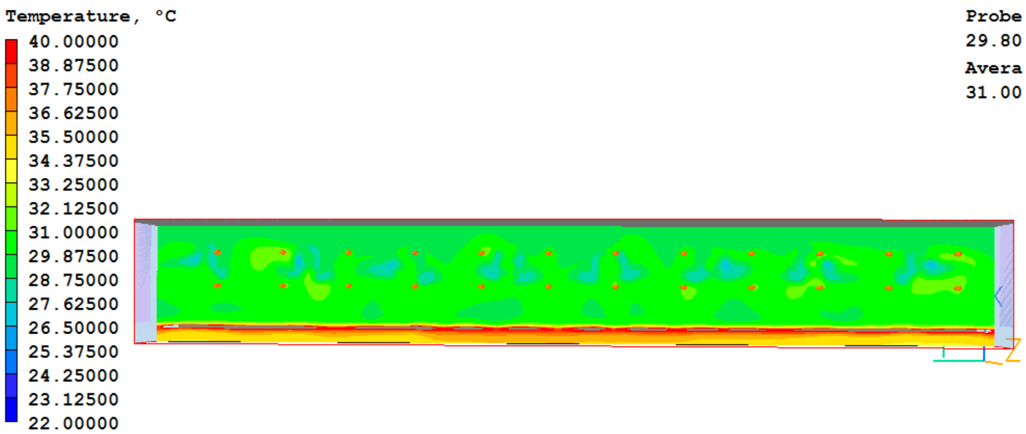

Figure 5: Temperature profile for air supply from ceiling – (option A)

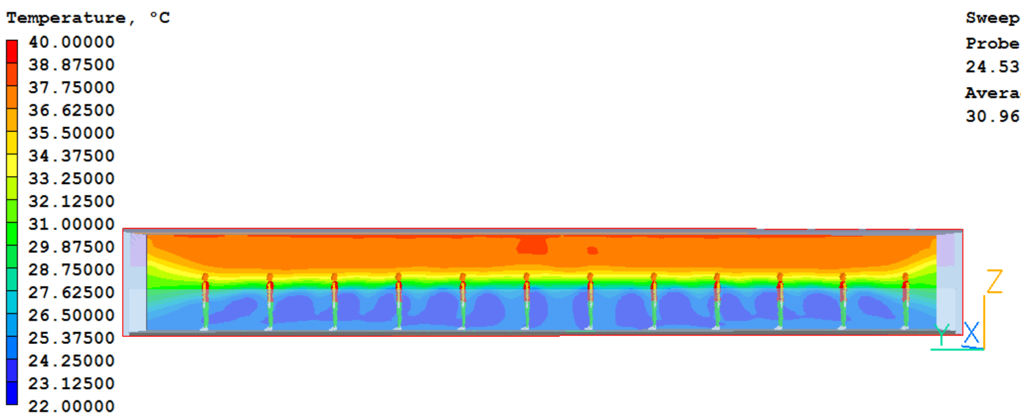

Figure 6: Temperature profile for air supply from ceiling - (option B)

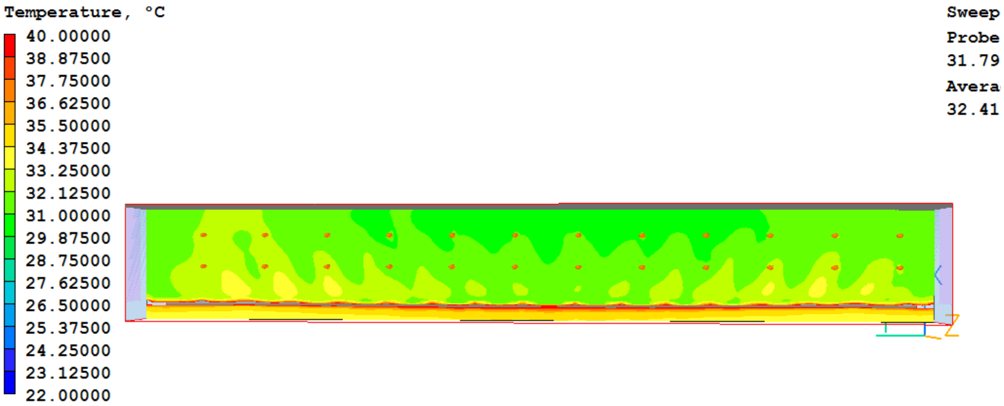

Figure 7: Temperature profile for air supply from floor level - (option C)

Not only the length section along the corridor, different parameters at cross sections and horizontal levels will be compared between different options.
Based on the barrier heights from the floor level, there are four sub-options for each option types of A, B and C which will be described in next section. For each option type, the sub-options have same grille arrangements (e.g. discharging angle, grille type) which have been firstly evaluated by various CFD simulations for the corridor.

4. Analysis on the simulation results

Average temperatures at critical points are compared for different options as in Table 1.

Table 1: Comparison of average temperature for different options

Options	Length section (<2 m)	Cross section (<2 m)	1.5 m high plane	1.1 m high plane
A1 (1.0m high barrier)	29.6	29.5	29.8	30.0
A2 (1.3m high barrier)	29.5	29.2	29.7	29.8
A3 (1.5m high barrier)	29.5	29.0	29.7	29.7
A4 (2.0m high barrier)	28.8	28.8	29.5	29.2
B1 (1.0m high barrier)	29.8	29.8	30.2	30.5
B2 (1.3m high barrier)	30.0	29.7	30.6	31.0
B3 (1.5m high barrier)	29.5	30.0	31.0	32.0
B4 (2.0m high barrier)	30.0	30.0	31.5	32.0
C1 (1.0m high barrier)	28.0	28.6	32.5	27.2
C2 (1.3m high barrier)	27.8	28.2	31.7	26.5
C3 (1.5m high barrier)	27.4	27.4	30.0	26.0
C4 (2.0m high barrier)	26.8	26.8	29.2	25.1

CFD simulation results show that the distribution of temperature and air flow is affected by the different settings (position, height and air flow angle) of the air supply, the return air, and, in addition to the air conditioning cooling load and the heat load and distribution in the half open corridor.

A total mesh size of 799,008 cells was used, non-uniformly distributed over the entire calculation domain. A converged solution was obtained after 6,000 iterations. Figure 2 and Figure 5 shown up the temperature distribution in the corridor during the summer for air supply from ceiling. As may be seen, the temperature in most areas is from around 25 °C to 30 °C, which is similarly shown in Figure 3 and Figure 6. The temperature contours superimposed by velocity vectors at the vertical section in the middle of the corridor (summer case). Figure 4 and 7 display the temperature distribution in the corridor during the summer for air supply from floor level.

Based on the CFD results, the general findings are summarized:

The proposed cooling load can meet the overall cooling requirements for the corridor. For a certain cooling load, the adjustable position and type of grille and the height of barrier along the corridor are the critical factors to be considered. It is necessary to adjust the grilles to achieve more even distribution of air flow.

- Air distribution pattern at horizontal levels or vertical sections are highly affected by the grille arrangements including the density, angle and discharging area.
- The cooling performance is better for the corridor with higher barrier under a certain level, especially for the case with low level air supply. But there is no significant improvement if the barrier is too high as 2m from the floor level, which will restrict the heat loss and air flow. Therefore, it is not recommended to increase the barrier too high.
- As the external hot air is easily be entrained into the corridor and restrict the air exchange, the scenarios B1 to B4 have the worst cooling performance.
- Scenarios C1 to C4 have the best cooling performance for lower level 1.1 m or below.
- Scenarios A1 to A4 have the best cooling performance for the higher level above 1.1 m.
- There is no great difference on the cooling performance between Case A and C, but most sensible area is the relatively higher level for passenger passing through the corridor, which is different to that siting in an office.

5. Conclusions

Temperature profile and air flow distribution at the corridor with various possible cooling options have been successfully studied by the CFD approach. The required thermal comfort for the concerned corridor could be achieved with cooling air of 2.4 m^3/s and 22 ^oC. Worst performance is found for the option with air supply from ceiling side with much hot air entrained and convection blockage. There will be the most energy saving for the cooling air supply from the floor level but greatest impact on the normal operation with much A&A works. For the air supply from ceiling centre, the thermal comfort is better for narrow grills in series along the corridor than that installed in parallel. Feasible cooling scheme should be proposed based on the cooling performance, air distribution, idea barrier height, energy saving, and operation impact. CFD model is a useful tool to support the design especially for any uncertain condition or the situation and geometry is too complicated.

References

Chow W.K., Gao Y., 2002, Numerical Simulation on the Wind Action and Fire Induced Flow Field of an International Finance Building in Beijing, Acta Aerodynamica Sinica, 20(3), 351-360.

Ren X., 2005, Numerical Simulation of Flow Field and Temperature Field in Air Conditioning Room. Dissertation for Master Degree, Harbin Engineering University, Harbin, China.

Liu C.M., Gao Y., 2012, Numerical Simulation of Airwake around Ski Jump Decks, Journal of Huazhong University of Science & Technology, 40(10), 68-71.

Gao Y., Liu C.M., 2013, Numerical Calculation of Frigate Ship Airwake, Journal of Harbin Institute of Technology, 34(5), 599-603.

Huo Y., Gao Y., 2013, Numerical Simulation of Thermal Fluid in Partially Open Enclosures Based on Vreman Subgrid-scale Model, Chinese Journal of Computational Mechanics, 30(1), 117-123.

Wang J.L, Gao Y.,2016, The Influence of Ship Tail Structure to Airwake on Helicopter Operation Area, Journal of Harbin Institute of Technology, 48(10), 148-154.

Xue R.J, Wang G., Fang L., 2005, Numerical Simulation of Thermal Comfort Degree in Air-conditioning Room, Journal of Harbin Institute of Technology, 37(8), 1145-1147.

Zhang Y.M, Zou G.W., Dong H., 2009, Predicting the Influence of Different Ventilation Arrangements on Thermal Comfort through FDS, Building Energy & Environment, 28(2), 148-154.