CFD Prediction of Contaminant Distribution in Indoor Car Park

Tang D [1], *Beattie K* [2]
[1] IES Limited, Glasgow, United Kingdom
[2] Dept. of Engineering, Dublin Institute of Technology, Dublin, Ireland

Introduction

The assessment of human exposure to airborne contaminant is an important issue in building design. The physiological significance of such exposure and technical means to minimise such risks have long been known in literatures. (1, 2, 3) In recent years, computational works have increasingly been seen used as design assessment tools as an alternative to site measurement and wind tunnel tests. Questions arise that most of such computation studies were restricted to single spaces due to high computing requirements and theoretical complexities ignoring the physical reality that the spaces were not isolated, it is encased by walls, occupants and furniture which were participating in the thermal environment. To address these issues two schools of thoughts have emerged. One was to enhance the CFD codes with the fabric and long-wave radiation models to enable it to model both airflow and thermal conduction simultaneously by prescribing boundary conditions external to the building. (4, 5, 6) The other was to extend the conduction based building thermal simulator to include the CFD codes so as to enable the two models interacting on a time-step basis, exchanging information at their model boundaries. (7, 8) Theoretically the two thoughts are sound with which the former was already be seen in rocket and turbine engine designs. While in CFD simulation of buildings they only succeeded in limited cases of academic studies. The main reason for this is that both the two approaches have been unable to handle the differences in time constants between the walls and the air volumes, which very often differ hundreds of times of order of magnitude. In the case of a realistic building, any attempt to simulate the two systems on a time-step basis would be rendered unnecessary.

In this paper an integrated approach is employed to take into account of the large differences in time constants between walls and air in the buildings, the CFD code is refined to account for the turbulent, buoyancy driven flow and the particle movement of contaminant.

Methods

The Integrated Thermal and CFD Simulation

Given that the large differences in time constants between the walls and the occupied air volumes within buildings, and furthermore, in CFD simulation each of the occupied space is further divided into tens of thousands of control volumes, the time constant of each of these control volumes will become negligible in comparison to those of the walls. Simultaneous or in time-step basis solution of the combined system would become unnecessary in practice. Special numerical treatment would be needed.

One solution to this is to divide the simulations into two different time frames, i.e. the large time constant and the short time constant frames. (9) In the first frame a comprehensive thermal simulation including the modelling of the wall conduction, solar, short wave and long wave radiation, casual gains with coupling to network based airflow and single node air volume simulation will be carried out for a longer period from several weeks to several months. The second frame looks into the transient airflow for any selected air volumes for a period of several minutes within the duration of the thermal simulation. This treatment is based on the concept that the transient volume-weighted averaging air temperature of the CFD simulation would be identical to the air temperature of the same space obtained from the thermal simulation. (10)

The CFD Numerical Model

The CFD software used for the prediction was developed by the author, which solves the 3D non-isothermal Navier-Stokes equations and species dispersion using finite difference discretisation. The special features of the code include, the RNG based k-ε turbulence model to account for the low Re commonly occurred inflow within buildings; the second-order QUICK algorithm for the discretisation of the convective terms; the SIMPLEC algorithm for the velocity-pressure coupling; and an transient implicit iterative solver for time-stepping.

The CFD model is built for the indoor car park within the building complex. Fumes and heat contaminant, CO, generated by the starting of the pared cars were modelled as heat and contaminant sources based on the monitored averaging daily frequency of the use. The geometry model was further divided into 98X by 75Y by 53Z cells for numerical simulation. Figure 1 and 2 show the geometry and the discretised models.

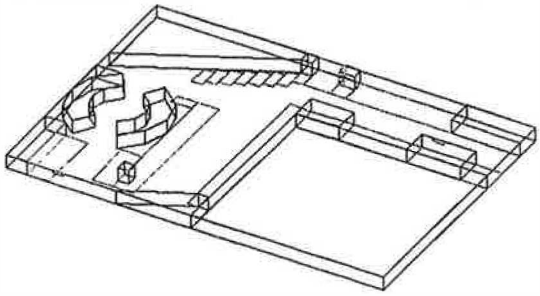

Figure 1.

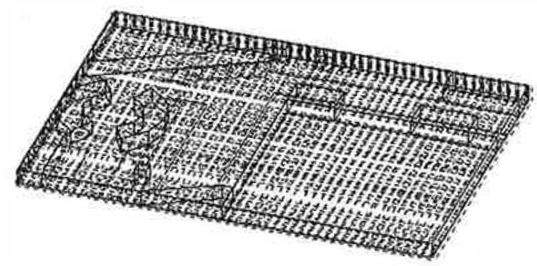

Figure 2.

Boundary Conditions

The large time constant frame thermal simulation for the whole building was carried using the energy simulation code, i.e. ESP, for a duration of four months starting January climate typical climate data of Edinburgh. (11) The boundary conditions required for CFD simulation were imported from the building thermal simulation. These had included the surface temperatures of the walls, floors and ceiling by which the effect of short and long-wave radiation among the surfaces had been taken into account; the convective contribution from lighting, occupants and equipment; airflow from the adjacent spaces. Thermal and momentum log-law wall functions were applied on all solid surfaces. For turbulence quantities k and ε the following relationships were applied at solid region:

$$k = \frac{u_*^2}{\sqrt{C_D}} \quad \varepsilon = \frac{u_*^3}{kz}$$

Parallel to this, in the CFD simulation, the convective heat transfer coefficients for the air-cell adjacent to the wall are updated at each time-step based upon the first principle theory.

Results and Discussion

Simulation results showed details of the air movement and distribution of contaminant where deficiency of the design of the system can be identified. The effectiveness of the ventilation system was assessed by investigating the concentration dispersion of the contaminant distribution. From the simulation the locations where the accumulation of higher concentration of contaminant were likely to occur were identified, which agreed with site measurement and based on these counter measures were considered to minimise the risk. To track the movement of contaminant, a first-order participating particle transport model based on particle mechanic was applied to calculate the trajectories of particles by considering their descending velocity. The results were shown in 3-D animation movement.

Conclusion

The combined building thermal and CFD simulation requires particular attention in handling the difference in time constant between the wall structures and the occupied air volumes, without which any attempt in tackling the problem would be rendered. To assess the CO pollution risk to the indoor car park using CFD simulation, a two-stage time frame simulation approach was proposed. In doing so, a seasonal thermal simulation was carried to model the long term thermal condition of the whole building and CFD simulations were then carried out using the boundary condition generated by the thermal simulation for short time frame analysis. The theoretical basis which incorporated advanced numerical necessary to the problem was introduced and the CFD model described. The CFD simulation results showed details of the contaminant distribution for locations where contaminant risk may occur were identified and compared with site measurements. Particle tracking algorithm was used to display the animation of contaminant dispersion.

References

1. AIVC, 1991. A Guide to Contaminant Removal Effectiveness, Air Infiltration and Ventilation Centre Technical note 28-2
2. Kimmel, C.A. & Gaylor, D.A. 1988. Issues in Quantitative and Qualitative Risk Analysis for Developmental Toxicology, Risk Analysis, Vol. 8, pp.15–20
3. CIBSE, 1988. *CIBSE Guide*. Xxxx
4. Holmes, M.J. Lam, J.K-W. Ruddick, K.G. & Whittle, G.E. 1990 'Computation of Conduction, Convection, and Radiation in the Perimeter Zone of an Office Space', *Proc. ROOMVENT '90*, Oslo Norway.

5. Chen, Q. Peng, X. and van Paassen, A.H.C. 1995. 'Prediction of Room Thermal Response by CFD Technique with Conjugate Heat Transfer and Radiation Models', *ASHRAE Transactions*, 3884 50–60.
6. Moser, A. Schalin, A. Off F. Yuan, X. 1995. 'Numerical Modelling of Heat Transfer by Radiation and Convection in an Atrium with Thermal Inertia', *ASHRAE Trans.* SD-95-14–4.
7. Clarke, J. Fraser, S. & Tang, D. 1990. An Investigation of the Functional Conflation of Building Energy Simulation and Computational Fluid Dynamics, EPSERC Grant Proposal, University of Strathclyde
8. Negra˜o C.O.R. (1995), *Conflation of Computational Fluid Dynamics and Building Thermal Simulation*, PhD Thesis, University of Strathclyde
9. Nielsen, P.V. & Tryggvason, T. 1998. Computational Fluid Dynamics And Building Energy Performance Simulation, *Proc. of RoomVent'98*, Stockholm Sweden
10. O'Brian, D. 1998. *QA Methodologies for Simulation of the Built Environment*, MSc Thesis, Department of Mechanical Engineering, University of Strathclyde
11. IES, 1997. *The IES<Virtual Environment> Product Summary*, www.ies4d.com, IES Limited.