A Review: Coupled Simulation of CFD and Network Model for Heat and Contaminant Transport in a Building

Shinsuke Kato*1 and Weirong Zhang2

1Professor, Institute of Industrial Science, the University of Tokyo, Japan
2Researcher, Institute of Industrial Science, the University of Tokyo, Japan

Abstract
In this study, the coupled simulation of CFD (Computational Fluid Dynamics) and network model of building energy (heat) and airflow simulation is reviewed. The network model is a tool to describe the whole and macroscopic characteristics of heat or airflow transport in buildings, where heat flow or airflow is modeled as a network of one-dimensional flow elements. It can be conducted with comparatively less computational cost. The temperature or contaminant concentration of the room air is often not uniform but spatially distributed in a room, when the mixing efficiency of room airflow is not high enough, or when sources and respective sinks of heat or contaminant are located comparatively apart in a room. The CFD is a tool to describe these distribution characteristics precisely. It requires considerable computation amount and computational time. It is not light or simple to do compared with the network model computation. If one wants to analyze the whole and macroscopic characteristics of heat or flow transports in buildings including the distribution features of temperature or contaminant in a certain room, or if one wants to analyze the room airflow with the consideration of whole building heat and flow transport, the two simulations should be done simultaneously. In other words, they should be solved with the coupled manner. There are many methods or ways for coupling the two simulations. In this review, the authors divided the coupled simulations into two groups. One is the group of which the CFD analysis is the main objective, while the other is the group with network simulation as the main objective. In the latter case, even the major objective is obtained from the results of the network simulation, and the full coupling CFD and network simulation requires an enormous amount of computation and time compared with the single network simulation. The full coupling is practically absurd and is sometimes impossible because of its enormous computation amount. Further simplification of the coupling with CFD has to be pursued. The authors have reviewed the papers published in the recent 10 years on this topic and classified relevant works into the two categories mentioned above.

Keywords: Computational Fluid Dynamics (CFD); Flow Network Model; Energy Simulation Model; Heat Transport; Contaminant Transport

1. Introduction
In a ventilated room, generated heat or contaminant is exhausted with exhaust airflow. The conservation law of mass or energy indicates that the generated heat or contaminant rate in a room is the same as the exhausted air from the room in the steady state condition. The room (volume) average temperature or contaminant concentration may not be the same as that of the exhausted air. If the source of heat or contaminant is located close to the exhaust opening and it is therefore quickly exhausted, the average temperature or concentration is almost the same as that in the supplied air from the supply opening and is different from that of the exhausted air. This fact is well known as the effectiveness of ventilation in a room. If a building engineer can utilize this phenomenon well, further energy saving can be achieved for the HVAC (Heating, Ventilation, Air-Conditioning) system in a building since the generated heat in the room is effectively exhausted. The ventilation effectiveness is the characteristics of complex three-dimensional airflow and it will be different from the point to point in a room (Kato and Murakami, 1988). There is an area where the ventilation effectiveness is comparatively good and an area which is vice versa. The temperature or contaminant concentration will be also three-dimensionally distributed and will vary with its boundary condition and source position. In other words, the characteristics of the ventilation effectiveness strongly depend on its...
The transport phenomena in buildings are usually three-dimensional in space and time-dependent. However, it is often applied for the building science that most of the transport phenomena are assumed to be time-dependent but one-dimensional for limited spaces or elements. Air flows in ducts or corridors or staircases are sometimes assumed to be unidirectional and thereby one-dimensional. Heat conduction by temperature difference between the two surfaces of plane shape building elements, such as walls, windows, ceilings, and floors, and so on, are sometimes assumed to be uniform and one-dimensional for the temperature difference. In this way, the whole heat transport and/or air movement phenomena in a building are sometimes able to be expressed as the assembly of one-dimensional transport elements between the many conjunction nodes in which heat or air flows are merged or divided. The assembly of one-dimensional flow elements between conjunction points is called a network model and can be analyzed as well as the electric currents in the complicated electric circuit.

For energy simulation purposes in a building, time-dependent heat transport is modeled as an electric circuit (network) where heat flow corresponds to electric current and temperature corresponds to voltage. The temperature difference causes the heat flows with a certain heat resistance (conductance). The dynamic features of heat flow are modeled with response function or solving time-dependent equation for one-dimensional heat resistance with a finite difference method. The time scale of the capacity effect in heat flow through heat resistance is usually big enough and slow; we cannot neglect it with the dynamic change of temperature. For airflow simulation purposes in a building, airflow are also modeled as an electric circuit (network) where airflow corresponds to electric current and static pressure corresponds to voltage. The static pressure difference causes airflow with a certain total pressure loss by flow resistance. The static pressure change in a building is usually small enough compared with the atmospheric pressure and the elastic features of air are very fast (sound propagation speed) compared with the air velocity in a building. The air is thereby assumed as incompressible and its density is only dependent on the temperature. If building components are stiff enough and the envelope volumes thereby do not vary with the change of static pressure, the capacity effect of airflow can be negligible. The dynamic features of airflows are usually modeled as instantaneous equilibrium and the airflows obey instantaneously with the change of static pressure. Since the treatments of the dynamic features of heat flows and airflow are different, the network model of heat transfer and airflows are different and separated. Figs. 1. and 2. indicate the network models of heat flows and airflows. Some examples of energy simulation models in buildings where heat flow network models are analyzed are "TRNSYS" (TRNSYS, 2000), "EnergyPlus" (Crawley et al., 2004),
and "ESP-r" (Jon, 2011). Some examples of airflow simulations in buildings are "COMIS" (Feustel, 1999), and "CONTAM" (Walton, 2005). Heat is transported by airflow and thereby both network models are often analyzed in the coupling manner.

2.1 Heat Transport by Airflow in a Room

In a room, heat is transported with air movement, i.e. convection and radiation. Here, the convective heat transport by airflow is considered. Later, the authors will consider the radiation heat transport. The example of the heat network model in a building with a large indoor space is shown in Fig.3.

In a large indoor space, the indoor airflow conveys the heat to and from the surface of the building elements such as walls, ceiling and floor. If the convective heat transport by the room air is fast enough compared to the heat transport through building elements in conduction with the delay due to heat capacity, the heat transport by room airflow can be assumed as in quasi-equilibrium state, i.e. as in the quasi-steady state. The room airflow delivers heat instantaneously from the vicinity of the building elements such as walls to the others in the room. It means that the vicinities of the building elements are connected with only the resistance due to the room airflow convection and without any capacitance. This situation can be represented with only one nodal point of the temperature in the room and each different convective heat transport resistance (conductance) to the vicinity of the building element. The nodal point should represent the amount of heat of the room air, i.e. the volume averaged room air temperature considering the conservation law of heat in a room. Around the indoor vicinity of the building elements, the heat transfer between the room air and the surface of building elements is also so fast that the delay of heat capacity effect can be neglected and the heat flow from the surface of the building elements to the vicinity room air can be modeled with the convective heat transfer coefficient for the building elements. The example of the network model with this assumption is shown in Fig.4.

Note: Heat conductance exists between any two nodes and is different due to the direction of heat flow. Here only the heat flows coming from/ going into node a and node b are illustrated in Fig.4.
One temperature nodal point of room air and the synthesized heat conductance which is composed with the convective heat transfer coefficient and the convective heat transport conductance to the nodal point for each building element will represent the delivery of heat by the room airflows. Most of the network models of heat flows in a building assume this quick heat transfer within a room and the effect of temperature distribution in the room for heat transfer to the building element is attributed to the different values of the synthesized convective heat transfer coefficient and airflow convection heat transport conductance for each building element. When the same synthesized value of the heat conductance by room airflow and the convective heat transfer coefficient is used for the building elements such as the ceiling, the floor, and walls, it means that perfectly well mixed room airflow and the uniform temperature distribution in the room is assumed. This situation is not true. However it is sometimes deserved as the first (rough) approximation and is practically proper when the room air is well mixed with the forced convection by mechanical ventilation or by air-conditioning.

The assumption of well mixed room airflows is not encountered in the room where heat sources and heat sinks are located apart or in the large indoor space where the buoyancy becomes strong. There will be some places where airflows become weak and strong. The mixing efficiency is not enough and the temperature is not uniform in the room. It is difficult to assume the proper convective heat transfer coefficient and proper airflow convection conductance for each building element. In this case, airflows and heat convection in a room can be predicted accurately by CFD. There will be two ways of utilization of CFD. One is the way in which CFD is used to evaluate the convection heat conductance by airflow and convective heat transfer coefficient both for each building element. In this case, if the convection conductance by airflow and convective heat transfer coefficient can be constant, only the network simulation is carried out and there is no need for coupling CFD for every time step of the network simulation. This situation is limited on the assumption that airflow properties are in a steady state and the delivery of heat by airflow is fast enough compared to the heat conductance within the building elements with conduction. The other way is the full coupling of CFD and network simulation. The CFD will obtain the boundary conditions from the network model at every time step of the simulation and the network simulation also obtains the heat flow and temperature conditions at the CFD boundaries. The full coupling will be done with the consideration of the heat conservation law.

With time-dependent CFD simulation, convective heat transfer from the solid surface to the room airflow and convection heat transport by airflow in the room can be predicted well, including dynamic delay with temperature change and temperature distribution in the room. Some examples of CFD are "Fluent" (Fluent, 2003), and "STAR-CCM+" (CD-adapco, 2009). CFD analysis can be done with a network simulation of building energy simulation. CFD analysis can be also done with a network simulation of airflow in buildings. Later, we will see the detailed reviews of coupling simulations of the CFD and network model.

2.2 Heat Transport by Radiation in a Room

In a normal room, the efficiency of heat transfer within the room by radiation is higher than that by convection. The radiation heat transfer is quick with light speed and always becomes equal for the heat exchange within a room's surfaces. From this point of view, the accuracy of radiation heat transfer within a room should be higher and more important than the convection heat transfer analysis. The radiation heat transfer in a room is often linearized and is modeled using radiation heat transfer coefficient and the mean radiation temperature. In the network simulation of heat flows in a building, sometimes the radiation heat transfer is analyzed by introducing the radiation heat transfer coefficient and only one nodal point for representing room air temperature. On the contrary, if CFD is coupled with heat transfer in the room, the accurate radiation analysis is inevitable for ensuring reasonably accurate heat transfer analysis.

Room airflow simulation with CFD is sometimes conducted with the airflow network model of a building with coupling manner. The network model is used to determine the inflow air volume and the outflow air volume rate. Except that the room air temperature is unknown and is obtained from CFD analysis of the room air and temperature distribution, it is not often that the CFD result affects the airflow network simulation in a building. Later, we will see the examples of the coupled simulation of CFD and airflow network model of buildings.

3. Two Groups of Coupling Simulation of CFD and Network Modeling of Heat Flows in Buildings

3.1 Simulation of the Whole and Macropscopic Characteristics of Building Energy Simulation Considering Temperature Distribution in a Room

In the network simulation of heat transport in a building, as stated in section 2.1, the temperature nodal point of the room which represents the amount of air heat in the room and the convection heat conductance by room airflow and convective heat transfer coefficient both for each building elements are required when the heat delivery characteristics of the room airflow is considered. If the convection conductance...
by room air flow is evaluated before the network simulation, there is no need for CFD coupling for
the network simulation of heat flow simulation in the building, even including the accurate effect of the three
dimensional properties of room air flow convection. This way does not require the CFD full coupling and
the computation load is almost the same or a bit higher with the conventional network simulation where
perfect mixing features of the room airflow is assumed.

Unfortunately, there are only a few researches in
this category. Kato (1994) has consistently pursued
this way. He has proposed CRI (Contribution Ratio
of Indoor Climate), which expresses the sensitivity of
the heat flux from the surface of the building elements
on the room air temperature at any point of concern
in a room. The volume integration of CRI means the
total contribution of the heat flux from the building
element on the room air. CRI is deduced from the
CFD simulation of room airflows in steady state and
the following three-dimensional temperature transport
simulation in also steady state condition. CRI does not
directly give the synthesized heat conductance of the
convection heat transport by airflow and convective
heat transfer coefficient from the building element.
It gives however the same degree of information of
heat conductance by airflow and convective heat transport by airflow and convective
heat transfer coefficient from the building element.
In this way does not require the CFD full coupling and
the computation load is almost the same or a bit higher
with the conventional network simulation where
perfect mixing features of the room airflow is assumed.

The same concept has been applied in transient
cases and is known as the response factor of heat
sources. Hiyama et al. (2010, 2011) developed a
calculation method that can achieve more accurate
time-series analysis. They calculated, in advance, the
heat response in a static flow field using computational
fluid dynamics (CFD) analysis. Then they calculated
Advection–Diffusion Response Factors and integrated
them into the energy simulation as a factor in the three-
dimensional thermal distribution within a room.

3.2 Simulation of Temperature Distribution in a
Room with Boundary Condition Evaluated from
Building Energy Simulation

Most of the coupled simulations conducted were in
the full coupling manner. CFD and the network model
are coupled in every time step or CFD coupling is not
conducted in every time step, but in the certain time
interval of network simulation since the airflow in a
room is not apparently changed.

A general building energy simulation needs an
accurate convective heat transfer coefficient and an
appropriate room reference temperature that can be
calculated by CFD. Conversely, CFD requires interior
heat boundary conditions that can be determined by
building energy simulation tools. Considering the
results provided by building energy simulations and
CFD programs, integrating the two programs for
indoor spaces with a non-uniform air temperature
distribution is considered as a possible solution.

Clarke et al. (1995) and Negrao (1998) implemented
an automatic airflow coupling with CFD inside the
Environmental Systems Performance Research (ESP-r)
with CFD. However, in their studies, different coupling
methods resulted in dramatically different results for
the same airflow problem, which indicated multiple
solutions of airflow coupling.

Bartak et al. (2002) refined the CFD module of
ESP-r and summarized the form of the CFD model,
described the method used to integrate the thermal
and flow domains and reported the outcome from an
empirical validation exercise.
Djunaedy et al. (2005a, 2005b) described the implementation of external coupling between building energy simulation and CFD. A summary of the latest development was presented to put external coupling into context. The implementation of external coupling was then summarized. Their work concluded with a case study to validate the performance of external coupling, and to highlight its potentials.

Zhai et al. (2002, 2003, 2004, 2005, and 2006) explored the principles, methodologies, strategies, implementation, and performance of energy simulation and CFD coupling. Their studies proved that the unique solution of coupling CFD and building energy simulation exists in theory, and they performed a sensitivity analysis of the coupling simulation. However, the direct coupling of CFD with energy simulations is still time-consuming. With the increasing numbers of variables, the coupling of CFD will cost more calculation resources. Thus, simulations will be limited to short periods or a limited number of time steps.

In order to account for the time varying thermal response of building fabrics to internal and external ambient conditions and the consequent effect on the air inside the enclosure, Somarathne (2005) proposed a DTM–CFD procedure utilizing a transient time-varying grid schedule, 'Freeze-Flow' and 'Boundary Freeze' techniques. 'Freeze Flow' paused the solution of all governing equations of fluid flow, except temperature; while 'Boundary Freeze' froze temperatures at boundaries of the CFD model whilst solving all equations in the flow domain.

In the study of Fan and Ito (2012), three types of ventilation systems were chosen for analysis of the coupling computational fluid dynamics (CFD) program with building energy simulation (BES) software, aiming to analyze the energy conservation performance of a real office space with an energy recovery ventilator (ERV) and to investigate the effect of the arrangement of air supply and exhaust openings.

4. CFD Coupled with Airflow Network Simulation in a Building

Most of the coupling simulations of CFD and network model of airflow in a building are the full coupling ones. The coupled simulation will be conducted only for a short term such as one hour or so. The coupling procedure with CFD and airflow network model was investigated well by Wang and Chen (2007a, 2007b and 2008). They found that the coupling with static pressure will work well compared to that with airflow rate. The pressure coupling method uses the static pressure as the given boundary conditions for the CFD and flow network model. With the given static pressure, the airflow rates at the boundary are solved when both the simulations and coupling iteration are conducted until the airflow rate of both the simulations coincide with the modifying static pressure at the boundary. A similar coupling method for the airflow rate can be modeled. With the given airflow rate, the static pressure at the boundary is solved when both the simulations and coupling iteration are conducted until the static pressure of both the simulations coincide.

The coupling method of the given airflow rate requires more coupling iteration times than that of the given static pressure method. The airflow is driven by the potential energy, i.e. the static pressure difference. The pressure difference is the cause of the airflow and the airflow is the result of the static pressure difference. The coupling methods might follow the natural principle of the cause and the result.

4.1 Smoke and Fire Simulation

The famous example of the coupling simulation is the fire simulation in a building. Fire Dynamics Simulator (FDS) is a CFD method for airflows in an enclosed space based on a large-eddy simulation (LES) for low-speed flows, with an emphasis on smoke and heat transport from fires (McGrattan et al., 2008). It can be done with the coupling manner of airflow network model CFAST (Consolidated model of Fire Growth And Smoke Transport) (Peacock et al., 2005). The success of these two simulations are well-known in the building science field and many CFD codes which enable the prediction of fire and smoke movement in a building have the option of coupled simulation with the major airflow network model simulations (Floyd, 2002, 2005, 2010; Yi et al., 2005; Lin et al., 2006; Shi et al., 2007; Cha, 2012).

A simulator coupled with various simulation programs named EVE SAYFA also has a similar function, in which the three-dimensional simulation, the Macro-Simulation and the Optimization for evacuation, etc. are connected with each other, referring to the Virtual Building. For the prevention or reduction of many kinds of disasters, the EVE SAYFA simulator aids their anticipation and the evaluation of safety (Nara et al., 2006; Ohba et al., 2008; Yamashita et al., 2009).

4.2 Contaminant Transport Simulation of Flow Network in a Building Coupled with Contaminant Distribution in a Room

An air contaminant transport simulation is important for controlling the indoor air quality in a building. Non-buoyancy air contaminant will not affect the airflow in a building. The analysis of the transport of non-buoyancy air contaminant in a building is different from the fire and smoke simulations. In an air-conditioned building, the feature of airflows in a building or in a room can be assumed as the steady state and the coupling simulations of CFD and the network model can be simplified as the steady state condition for airflows and only the contaminant transport is assumed as time dependent. The direct coupling of three-dimensional contaminant diffusion simulation in the room and the contaminant transport simulation in the building by network model is, of course, easy to conduct with the coupling method deduced from the fire and smoke simulation (Kondo et al., 2009).
The idea of coupling of multizone and CFD in building airflow simulations was proposed fairly early. Schaelin et al. (1992, 1993) proposed a method called method of detailed flow path values, in which, the perfect mixing assumptions of multizone models were remedied providing detailed pressures, velocities, and contaminant concentrations of flow paths from CFD. However, this study only implemented the coupling of contaminant concentrations and did not actually perform the coupling in airflow simulations. The coupling procedure was also an external or manual iteration.

Gao and Chen (2003), and Wang and Chen (2007a, 2007b and 2008) applied the coupled model of network model and CFD to calculate airflow and contaminant dispersion in a three-story, naturally ventilated building with a large atrium, assuming that a contaminant was released in the atrium. They also indicated that the best coupling method is pressure–pressure coupling that exchanges pressure between the multizone and CFD because it is most stable and can always lead to a converged solution. Numerical tests were further performed to verify the theory and it demonstrated that the coupled program is able to effectively improve the accuracy of the results.

The transportation of passive contaminant is a linear physical process and can be expressed with the response functions and convolution commutation. The meaning of "passive" is that a contaminant does not affect airflow and its transportation is completely controlled with the airflow. In this context, if people do not need the properties of whole contaminant distribution in a room but only want to understand the contaminant concentration at a certain point of concern, the contaminant transportation features can be expressed with response function for required or limited points in the room. The response function of the point in the room should be calculated from the three-dimensional time dependent simulation of contaminant diffusion in the room. Ishida and Kato (2008), Hiyama et al. (2008) have developed this method. The coupling method with network model is deduced in pure network simulation with the transportation in a room being expressed by the response function.

4.3 Energy Simulation of the Building Utilizing Natural Wind Induced Cross Ventilation

These days, utilization of natural ventilation such as wind induced cross ventilation and buoyancy driven ventilation become popular for controlling the thermal environment in rooms and saving the energy for mechanical air-conditioning. In the simulation, airflows and heat energy flows in buildings are simulated with an airflow and heat flow network model. In certain rooms, air and temperature distributions are simulated with CFD (Chang et al., 2004; Wang et al., 2008, 2009; Zhang et al., 2013). Both features of the whole building and a certain room are tightly related and are solved with the coupled manner. The coupled manner would be almost the same with fire and smoke simulation. Most of the researches conducted in recent decades were the full coupling manner and seemed to require huge computation amount. We, the authors, are a bit sorry that they use the full coupling manner and we would like to recommend using only network simulation including the heat and air transport features with the aid of CFD in the room.

5. Conclusions

The coupled simulation of CFD (Computational Fluid Dynamics) and network model in building energy (heat) and airflow simulation is reviewed. In this review, the authors have divided the coupled simulations into two groups. One is the group of CFD analysis being the main objective and full or semi-full coupling is conducted. Another is the group of network simulations being the main objective. The properties of three-dimensional transportation of heat or contaminant can be expressed with the response functions or simple conductance by airflow transportation. If the major objective is obtained from the results of the network simulation, the full coupling CFD and network simulation requires an enormous amount of computation and time compared with the single network simulation. The full coupling is practically absurd and is sometimes impossible because of its enormous computation amount. The simplification of the coupling with CFD is pursued from the viewpoint of the network simulation modeling.

References

1) Bartak, M., Beausoleil-Morrison, I., Clarke, J. A., Denev, J., Drkal, F., Lain, & Stankov, P. (2002). Integrating CFD and building simulation. Building and Environment, 37(8), pp.865-871.
2) Djunaedy, H., Hensen, J. L. M., & Loomans, M. G. L. C. (2005a). External coupling between CFD and energy simulation: implementation and validation. ASHRAE Transactions, 111(1), pp.612-624.
3) Djunaedy, E. (2005b). External coupling between building energy simulation and computational fluid dynamics. Ph.D. thesis. Technische University of Eindhoven.
4) CD-adapco. U. S. E. R. (2009). GUIDE: STAR-CCM+. 5) Cha, M., Han, S., Lee, J., & Choi, B. (2012). A virtual reality based fire training simulator integrated with fire dynamics data. Fire Safety Journal, 50, pp.12-24.
6) Chang, H., Kato, S., & Chikamoto, T. (2004). Effects of outdoor air conditions on hybrid air conditioning based on task/ambient strategy with natural and mechanical ventilation in office buildings. Building and environment, 39(2), pp.153-164.
7) Clarke, J. A., Hensen, J. L. M., & Negrao, C. O. R. (1995). Predicting Indoor Airflow by Combining Network Approach, CFD and Thermal Insulation. In DOCUMENT-AIR INFEILTRATION CENTRE AIC PROC (pp.145-145). OSCAR FABER PLC.
8) Crawley, D. B., Lawrie, L. K., Pedersen, C. O., Winkelmann, F. C., Witte, M. J., Strand, R. K., and Gu, L. (2004). EnergyPlus: new, capable, and linked. Journal of Architectural and Planning Research, 21(4), pp.292-302.
9) EVE SAYFA: Simulation System for Urban Safety and Environment.http://venus.iis.u-tokyo.ac.jp/english/Research/field-e/eyesayfa/
10) Fan, Y., & Ito, K. (2012). Energy consumption analysis intended for real office space with energy recovery ventilator by integrating BES and CFD approaches. Building and Environment, 52, pp.57-67.
11) Feustel, H. E. (1999). COMIS—an international multizone airflow and contaminant transport model. Energy and Buildings, 30(1), pp.3-18.
12) Floyd, J. (2002). Comparison of CFAST and FDS for Fire Simulation with the HDR T51 and T52 Tests. US Department of Commerce, Technology Administration, National Institute of Standards and Technology.

13) Floyd, J. E., Hunt, S. P., Williams, F. W., & Tatem, P. A. (2005). A network fire model for the simulation of fire growth and smoke spread in multiple compartments with complex ventilation. Journal of fire protection engineering, 15(3), 199-229.

14) Floyd, J. (2010). Coupling a network HVAC model to a computational fluid dynamics model using large eddy simulation. In Interflam 2010: Proceedings of the twelfth international conference, University of Nottingham, United Kingdom.

15) Fluent, I. N. C. (2003). FLUENT 6.1 User's Guide. Lebanon (NH), Fluent Inc.

16) Gao, Y., & Chen, Q. (2003). Coupling of a multi-zone airflow analysis program with a computational fluid dynamics program for indoor air quality studies. In Proceedings of the 4th International Symposium on HVAC, Symposium on HVAC (Vol. 1, pp.236-242).

17) Hiyama, K., Kato, S., Takahashi T., Huang H., Kobayashi, S., Iwase S. (2005). Building Design Utilizing Natural Ventilation Based on CFD and Ventilation Network Analysis and Its Performance Evaluation by Field Measurement, The International Journal of Ventilation, Vol.4, pp.13-24.

18) Hiyama, K., Ishida Y., Kato S., (2008). Method for Coupling Three-Dimensional Transient Pollutant Transport into One-Dimensional Transport Simulation Based on Concentration Response Factor, ASHRAE Transactions, Vol. 114.

19) Ishida Y. and Kato S., (2008). Coupled Flow Network Model and 3D Transient Pollutant Transport in a Room Using Concentration Response Factor Method, ASHRAE Transactions, Vol. 114.

20) Hiyama, K., & Kato, S. (2011). Integration of three-dimensional CFD results into energy simulations utilizing an Advection-Diffusion Response Factor. Energy and Buildings, 43(10), pp.2752-2759.

21) Ishida Y. and Kato S., (2008). A simple two-layer zone model on mechanical exhaust in an atrium. Building and Environment, 43(5), pp.348-361.

22) Lin, C. S., Wang, S. C., Huang, B., & Hsu, J. H. (2006). Ventilation effect on fire smoke transport in a townhouse building. Heat Transfer—Asian Research, 35(6), pp.387-401.

23) Matsumoto, T., Kato, S., Sasamoto, T., & Omori, T. (2008). PROPOSAL OF COUPLING CFD WITH ENERGY SIMULATION BY CRI. Fundamental research on the coupled simulation of CFD and energy simulation (Part 1). Journal of Environmental Engineering (Transaction of AII), 73(626), pp.445-450.

24) McGrattan, K. B., Hostikka, S., Floyd, J. E., Baum, H. R., & Rehm, R. G. (2008). Fire dynamics simulator (version 5), technical reference guide. NIST special publication, 1018, p.5.

25) Nara, M., Kato, S., Huang, H., & Zhu, S. W. (2006). Numerical analysis of fire on disasters with “EVE SAYFA”, a fire simulator, Assessing contribution of heat sources and sinks to temperature numerical simulation, ASHRAE Transactions, Vol.94, pp.309-330.

26) Negrão, C. O. (1993). Improvement of multi-zone model predictions by detailed flow path values from CFD calculation, ASHRAE transaction, 93-7-4, pp.709-720.

27) Shi, C., Lu, W. Z., Chow, W. K., & Hsu, J. H. (2007). An investigation on spill plume development and natural filling in large full-scale atrium under retail shop fire. International Journal of heat and mass transfer, 50(3), pp.513-529.

28) Somarathne, S., Seymour, M., & Kolokotroni, M. (2005). Dynamic thermal CFD simulation of a typical office by efficient transient solution methods. Building and environment, 40(7), pp.887-896.

29) Tan, G., & Glicksman, L. R. (2005). Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. Energy and Buildings, 37(10), pp.1049-1057.

30) TRNSYS, A. (2000). Transient System Simulation Program. University of Wisconsin.

31) Wang, L., & Chen, Q. (2007a). Theoretical and numerical studies of coupling multizone and CFD models for building air distribution simulations. Indoor Air, 17(5), pp.348-361.

32) Wang, L., & Chen, Q. (2007b). Validation of a coupled multizone-CFD program for building airflow and contaminant transport simulations. HVAC&R Research, 13(2), pp.267-281.

33) Wang, L., & Chen, Q. (2008). Applications of a coupled multizone-CFD model to calculate airflow and contaminant dispersion in built environments for emergency management. HVAC&R Research, 14(6), pp.925-939.

34) Wang, L., & Wong, N. H. (2008). Simulations for naturally ventilated residential buildings. Automation in Construction, 17(4), pp.386-398.

35) Wang, L., & Wong, N. H. (2009). Coupled simulations for naturally ventilated rooms between building simulation (BS) and computational fluid dynamics (CFD) for better prediction of indoor thermal environment. Building and Environment, 44(1), pp.95-112.

36) Walton, G. N., & Dols, W. S. (2005). CONTAM 2.4 user guide and program documentation. National Institute of Standards and Technology, NISTIR, 7251.

37) Yamashita, T., Soda, S., & Noda, I. (2009). Evacuation planning assist system with network model-based pedestrian simulator. In Principles of Practice in Multi-Agent Systems (pp.649-656). Springer Berlin Heidelberg.

38) Yi, L., Chow, W. K., Li, Y. Z., & Hsu, H. (2005). A simple two-layer zone model on mechanical exhaust in an atrium. Building and environment, 40(7), pp.869-880.

39) Zhai, Z., Chen, Q., Haves, P., & Klems, J. H. (2002). On approaches to couple energy simulation and computational fluid dynamics programs. Building and Environment, 37(8), pp.857-864.

40) Zhai, Z., & Chen, Q. Y. (2003). Solution characters of iterative coupling between energy simulation and CFD programs. Energy and Buildings, 35(5), pp.493-505.

41) Zhai, Z., & Chen, Q. (2004). Numerical determination and treatment of convective heat transfer coefficient in the coupled building energy and CFD simulation. Building and Environment, 39(8), pp.1001-1009.

42) Zhai, Z., & Chen, Q. Y. (2005). Performance of coupled building energy and CFD simulations. Energy and buildings, 37(4), pp.333-344.

43) Zhai, Z. J., & Chen, Q. Y. (2006). Sensitivity analysis and application guides for integrated building energy and CFD simulation. Energy and Buildings, 38(9), pp.1060-1068.

44) Zhang, R., Lam, K. P., Yao, S. C., & Zhang, Y. (2013). Coupled EnergyPlus and computational fluid dynamics simulation for natural ventilation. Building and Environment, 68, pp.100-113.

45) Zhang, W., Kato, S. (2012). Study on Characteristics of Indoor Heat Transfer in Non-Air Conditioned Environments, Proceedings of IBPC2012, Kyoto.

46) Zhang, W., Hiyama, K., Kato, S., & Ishida, Y. (2013). Building energy simulation considering spatial temperature distribution for nonuniform indoor environment. Building and Environment, Vol.63, pp.89-96.