Building and Environment 63 (2013) 89–96

Contents lists available at SciVerse ScienceDirect

Building and Environment

journal homepage: www.elsevier.com/locate/buildenv

Building energy simulation considering spatial temperature distribution for nonuniform indoor environment

Weirong Zhang*, Kyosuke Hiyama, Shinsuke Kato, Yoshihiro Ishida

Institute of Industrial Science, University of Tokyo, Tokyo, Japan

ARTICLE INFO

Article history:
Received 10 October 2012
Received in revised form
8 February 2013
Accepted 9 February 2013

Keywords:
Building energy simulation
Temperature distribution
CFD
Network model
Contribution ratio of indoor climate (CRI)

ABSTRACT

Building energy simulations are mostly implemented using network or multi-zone models that do not consider indoor air temperature and velocity distribution. However, the recent introduction of personal ventilation, floor-heating systems, and displacement ventilation positively utilizes a nonuniform indoor environment to meet the demand for both energy efficiency and thermal comfort. Therefore, it is expected that building energy simulation results will be significantly impacted by the choice of an appropriate reference air temperature for the calculation of heat transfer through constructed materials. This means that the air temperature distribution needs to be taken into account for nonuniform environments when carrying out energy simulations. An alternative approach to this problem is to combine a computational fluid dynamics (CFD) simulation directly with a network model; however, this approach is unfortunately too computationally time-consuming. In this study, we propose an acceptably fast simulation method that couples the contribution ratio of indoor climate (CRI), which is extracted from CFD results and indicates the individual impact of all heat factors, with the network model to implement an energy simulation that incorporates a temperature distribution. With the introduction of CRI, it is possible to achieve a precision as high as that of CFD and a calculation speed as fast as that of the network model. A case study simulating the thermal load of a single office room was carried out with the CRI-coupled method. The energy demand result calculated by CRI-coupled method was 15–20% lower than that of a non-coupled network simulation.

1. Introduction

Generally, energy simulation tools that simulate the energy performance of a building are based on a network model [1–3]; these tools include energy simulation programs like EnergyPlus or TRNSYS. In network models, it is assumed that the air in an indoor space is well mixed and that the temperature of the room or zone is uniform; this allows us to treat the rooms and zones as interconnected one-dimensional nodes. This method is generally called macro-simulation. Although a three-dimensional distribution of temperature and velocity exists, the error caused by treating it as the average value is highly tolerable. Therefore, the calculation speed can be vastly improved and an entire year's energy consumption can be assessed.

However, systems providing a nonuniform indoor temperature distribution have been widely introduced because they make positive use of the air current and temperature distribution to efficiently control the air temperature for the necessary space only, rather than keeping the temperature constant and uniform throughout the whole space. Such systems include task ambient air-conditioning systems, personal air-conditioning systems, displacement ventilation, and floor heating systems. It is expected that building energy simulation results will be significantly impacted by the choice of an appropriate reference air temperature for the calculation of heat transfer through construction materials. As a result, it has become necessary to include temperature distribution in energy simulations.

Computational fluid dynamics (CFD) programs can provide detailed distributions of air current and temperature, and they are usually used for predicting indoor thermal comfort and assessing indoor air quality [4]. Theoretically, CFD programs that accurately describe the heat transfer in solid materials and calculate the air velocity and temperature distribution can even predict the energy demand if all of the heat factors are set as dynamic boundary conditions and heat generators. However, this method is extremely computationally expensive and almost impossible to perform for a long-period energy simulation (i.e. a season or a year), even though

* Corresponding author. Tel.: +81 03 5452 6431; fax: +81 03 5452 6432.
E-mail addresses: zhangweirong129@gmail.com, zhangwr@iis.u-tokyo.ac.jp (W. Zhang).

0360-1323/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.buildenv.2013.02.007

the calculation speed of supercomputers has increased greatly over the years.

Thus, 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 nonuniform air temperature distribution is considered as a possible solution [5–7]. Zhai and Chen [8–11] 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 this study, instead of using direct coupling, we wish to present an alternative way of integrating CFD and a building energy simulation to analyze the energy demand of an indoor space with a nonuniform temperature distribution. This paper utilizes an index called the contribution ratio of indoor climate (CRI), which is extracted from and can thus present CFD simulation results. The CRI of each heat factor in the whole space indicates the contribution of that heat factor to the temperature distribution. By coupling CRI with the network model, the energy simulation can be achieved within the defined accuracy of CFD, but with a much-reduced calculation load. It can thus increase the accuracy of energy consumption predictions.

The concept and definition of CRI will be introduced first; then, the coupling strategy with the building energy simulation will be explained in detail. Finally, to demonstrate how to apply the proposed method, we will present a case study using the CRI-coupled method to simulate the thermal load of a single office room. The energy demand result calculated by CRI-coupled method was 15–20% lower than that of a non-coupled network simulation.

2. Contribution ratio of indoor climate (CRI)

2.1. Assumption of linear heat transfer in fixed airflow field

The indoor thermal environment is affected by various heat factors from both outdoors and indoors. The heat factors mentioned here include heat transfer through solid construction materials, warm or cool air supplies from air-conditioning systems, hot or cold radiant panels, ventilation or infiltration, and heat generation, such as heat from lighting, equipment, and the human body. Although these heat factors each have their own heat transfer characteristics, they all affect the air temperature either upward or downward by means of convection. This means that if the convective heat of all heat factors and their influence on the temperature field are known, the temperature distribution can be predicted.

In this study, all of these heat factors are treated and modeled as one of the heat source terms of the energy governing equation. One may question why heat transfer through constructions and air supply are generally treated as boundary conditions in the energy governing equation. Actually, the two methods are equal to each other in numerical calculations. For example, as shown in Fig. 1, the effect of the heat transferred through the solid wall is equal to that of the heat q [W/m^3] generated numerically from the first cells next to an adiabatic wall. Further, as shown in Fig. 2, the general

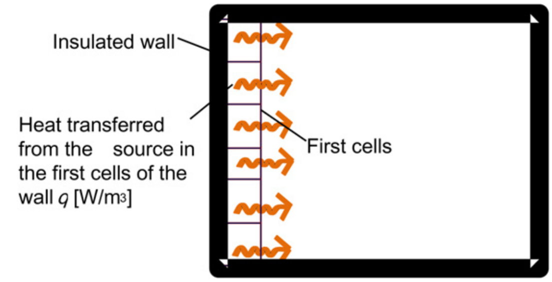

Fig. 1. Model of heat transfer through construction materials.

boundary condition of warm (or cool) air supply from a supply opening is separated into two parts. One is the non-warm (or non-cool) air supply condition at the boundary and the other is the imposed corresponding heat source, which raises (or decreases) the supply-air temperature in the governing equation.

Hence, in this study, all of the thermal boundary conditions in normal cases are assumed as source terms in the energy governing equation. The indoor temperature distribution can be seen as the temporal and spatial synthesis of the influence of all the heat sources, as expressed in Eq. (1):

$$\frac{\partial \theta}{\partial t} + \frac{\partial \theta u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{Pr_t}\frac{\partial \theta}{\partial x_j}\right) + \frac{q}{C_p \rho} \tag{1}$$

where θ is the air temperature, u_j is the air velocity, ν_t is the turbulent viscosity, q is the heat source, t is the time, x_j is the coordinate, Pr_t is the turbulent Prandtl number, C_p is the specific heat of air, and ρ is the air density.

Generally, the indoor air velocity field is determined by both forced convection and natural convection. When the airflow field is mainly determined by forced convection, as is the case in mechanical ventilation and air-conditioning systems, the airflow field can be considered fixed for a small variation range of the inlet air temperature and velocity [12]. If the buoyancy with density variation caused by temperature changes can be considered to have little effect on air motion, then it can be assumed that the temperature field in a room is a linear system. Based on this essential assumption, the temperature field expressed in Eq. (2) can be seen as the superimposition of several individual temperature fields, each of which is dominated by a single heat source expressed by Eq. (3):

$$\frac{\partial \Delta\theta_m}{\partial t} + \frac{\partial \Delta\theta_m u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{Pr_t}\frac{\partial \Delta\theta_m}{\partial x_j}\right) + \frac{q_m}{C_p \rho} \tag{2}$$

$$\frac{\partial \theta}{\partial t} + \frac{\partial \theta u_j}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t}{Pr_t}\frac{\partial \theta}{\partial x_j}\right) + \sum_m \left(\frac{q_m}{C_p \rho}\right) \tag{3}$$

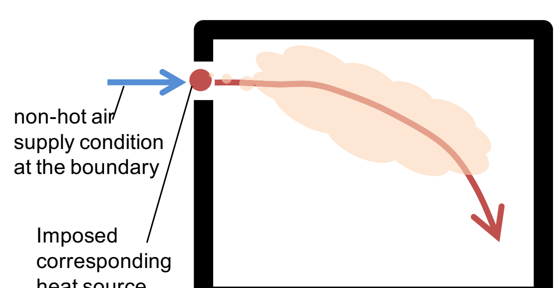

Fig. 2. Model of warm air supply.

where $\Delta\theta_m$ is the temperature rise caused by heat source m, and q_m is the heat generation of heat source m.

For instance, as illustrated in Fig. 3, the indoor environment of this room is affected by three heat sources: the heat supplied from the inlet, the heat generated from the human body, and the heat transferred through the walls. When the airflow field is fixed, the temperature field can be regarded as linear. Then the problem becomes how heat from each source transfers within the fixed representative airflow field. The temperature distribution caused by each of the three independent heat sources is calculated in the fixed representative airflow field to obtain the temperature subfields (a, b and c). Because of the linearity of heat transfer, the superimposition of these subfields is equivalent to the real temperature field (d).

2.2. Definition of CRI

The index Contribution Ratio of Indoor Climate (known as CRI) was originally proposed to evaluate how each single heat factor contributes to the indoor temperature distribution, by considering them separately [13]. The concept of CRI has since been extended and has spread to the evaluation of moisture and contaminant distribution [14–17]. The same concept has been applied in transient cases and is known as the response factor of heat sources [18,19].

CRI was previously defined as the ratio of the temperature rise (or decrease) caused by one individual heat source (or sink) at a location to the absolute value of the temperature rise (or decrease) caused by the same heat source with uniform distribution of the same amount of heat. It indicates how far the generated heat diffuses in the space. In other words, it indicates the range and the degree of influence of each heat source, and its value is not an absolute intensity (i.e., it is normalized by the value for its own perfect mixing condition). In places where the ratio is less than one, the influence of the heat source is smaller than in the case of perfect mixing. The CRI of the heat factor m at the location x_i is defined as:

$$\mathrm{CRI}_m(x_i) = \frac{\Delta\theta_\mathrm{m}(x_i)}{\Delta\theta_\mathrm{m,0}} = \frac{\theta_\mathrm{m}(x_i)-\theta_\mathrm{n}}{\theta_\mathrm{m,0}-\theta_\mathrm{n}} = \frac{\theta_\mathrm{m}(x_i)-\theta_\mathrm{n}}{(Q_\mathrm{m}/C_p\rho F)} \tag{4}$$

where x_i is the component of the spatial coordinates ($i = 1, 2, 3$), θ_n [°C] is the neutral temperature of a room (which has no physical meaning; it is used only for calculating CRI); $\theta_\mathrm{m,0}$ [°C] is the temperature of the room when the heat transfer Q_m from heat factor m is diffused uniformly; $\Delta\theta_\mathrm{m,0} = \theta_\mathrm{m,0} - \theta_\mathrm{n}$ is the temperature rise of the room from θ_n; $\theta_\mathrm{m}(x_i)$ [°C] is the temperature at position x_i caused by heat factor m as calculated by CFD; $\Delta\theta_\mathrm{m}(x_i) = \theta_\mathrm{m}(x_i) - \theta_\mathrm{n}$ is the temperature rise from θ_n of position (x_i) caused by heat source m; $Q_\mathrm{m} = \int_{V_\mathrm{Room}} q_\mathrm{m}\,dV$ [W] is the integral convective heat transfer from factor m; C_p [J/kg·K] is the specific heat of indoor air; ρ [kg/m^3] is the air density; and F [m^3/s] is the volume of supply air.

According to the principles of CRI, it must be assured that each temperature subfield is dominated by the heat source observed, without any effect from other heat sources. Thus, when applying Eq. (4) to natural convection, a steady temperature distribution cannot be obtained, because there is no heat sink to meet the total heat balance when only setting one heat source in the representative airflow field. In forced convection, heated or cooled air is exhausted by ventilation, which takes the generated heat out of the room. Thus, we propose to set a virtual heat sink, the heat of which counteracts the heat from the heat source, while the heat sink is set to have a uniform distribution across the space to avoid forming its own temperature distribution. The CRI of heat source m at location x_i was originally defined in a previous study [20,21] and is given as Eq. (5) below. It is the ratio of the temperature rise from the neutral temperature θ_n to the heat Q_m released from heat source m:

$$^u\mathrm{CRI}_\mathrm{m}(x_i) = \frac{^u\theta_\mathrm{m}(x_i)-\theta_\mathrm{n}}{Q_\mathrm{m}} \tag{5}$$

where $^u\theta_\mathrm{m}(x_i)$ [°C] is the temperature at position x_i calculated by CFD when the heat source Q_m and the heat sink are both set; the subscript u means a uniform heat sink is set in this case.

2.3. Using CRI to predict the temperature distribution

The CRI of each heat source to the temperature distribution can be seen as constant because the airflow field is fixed. That means that if the heat from one heat source increases by a factor of 3, the temperature rise anywhere will increase by a factor of 3. Thus, CRI can be helpful in calculating the dynamic temperature distribution. When the heat from any heat source changes, the temperature change caused by this heat source can be calculated at any location by multiplying the heat by its CRI. A new temperature distribution can then be calculated by superimposing the effect of all the heat

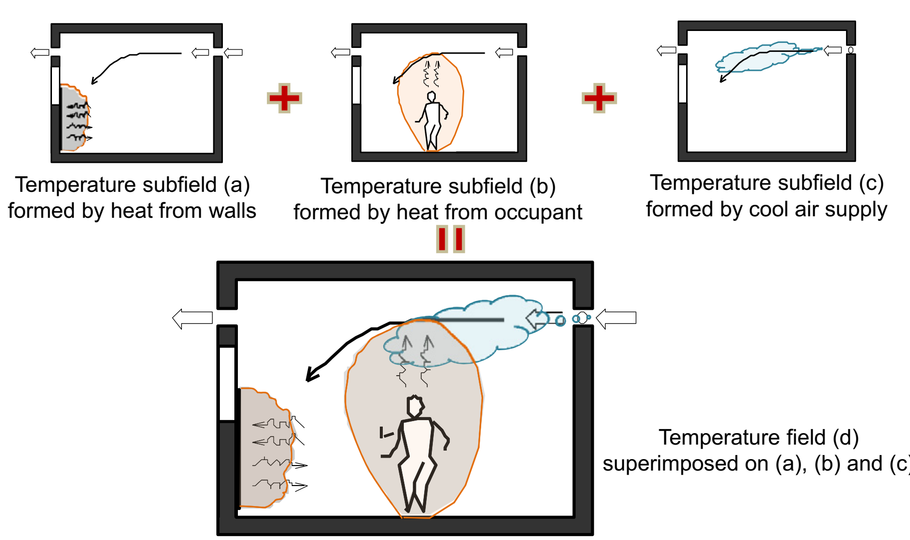

Fig. 3. Linear synthesis of temperature field in room with three heat sources.

sources. In the case of forced convection, the temperature distribution can be calculated by Eq. (6), which is derived from Eq. (4). For the cases of no air-conditioning or ventilation, the relation can be presented as Eq. (5), which is derived from Eq. (2).

$$\theta(x_i) - \theta_n = \sum_m \left(\mathrm{CRI}_m(x_i) \cdot \Delta\theta_{m,0}\right) \tag{6}$$

$$\theta(x_i) - \theta_0 = \sum_m \left({}^u\mathrm{CRI}_m(x_i) \cdot Q_m\right) \tag{7}$$

The CRI method has been validated by comparing calculation results with measurement data; a good correlation was obtained in previous study [14]. In this study, we will focus on how to couple the CRI to a building energy simulation.

3. Methodology

In a general building energy simulation, heat transfer through construction materials is calculated from the reference room temperature under the perfect mixing assumption. The main idea of the current study is to pair the temperature distribution calculated by CRI with the energy simulation, such that heat transfer can be calculated using the air temperature in the vicinity of the wall. This can be very helpful in non-uniform indoor environments. The CRI model will provide a dynamic temperature distribution, which is calculated based on the transient value of convective heat transfer of all the heat sources; the energy simulation tool will provide this transient convective heat transfer value based on a calculation with a proper spatial reference temperature.

There are several possible coupling strategies when combining the energy simulation and the CRI model, including full-dynamic and quasi-dynamic coupling [8]. In the former, an iterative calculation between the two models is carried out at every time step; this has a high precision, but is very time-consuming. In the latter coupling strategy, the information exchange between the two models is only carried out explicitly to reduce the time and computing consumption, which is acceptable when there is no violent change of boundary conditions between two time steps, such as in an office room.

Quasi-dynamic coupling is chosen as the coupling strategy in this study to meet the requirement of efficiency, as shown in Fig. 4. At time step k, the convective heat transfer from all the heat sources Q_m^k is used to calculate the temperature distribution $\theta(x_i)^k$ in the CRI model. Then $\theta(x_i)^k$ is used as the temperature distribution at time step $k+1$ to calculate the heat transfer of the walls Q_m^{k+1} at time step $k+1$.

The flow chart of the proposed coupling simulation is shown in Fig. 5. First, a radiation-coupled CFD simulation is carried out using the representative boundary conditions, by which the representative airflow fields can be obtained. The representative airflow field should include reasonable settings for all of the heat factors. In addition, the number of representative fields can be set to more than one to improve the precision of the modeling; for example, the field could be set to two separate fields, one with air-conditioning and one without air-conditioning. The more airflow fields that are set, the more precise the results that can be expected, and the more load calculations will be triggered.

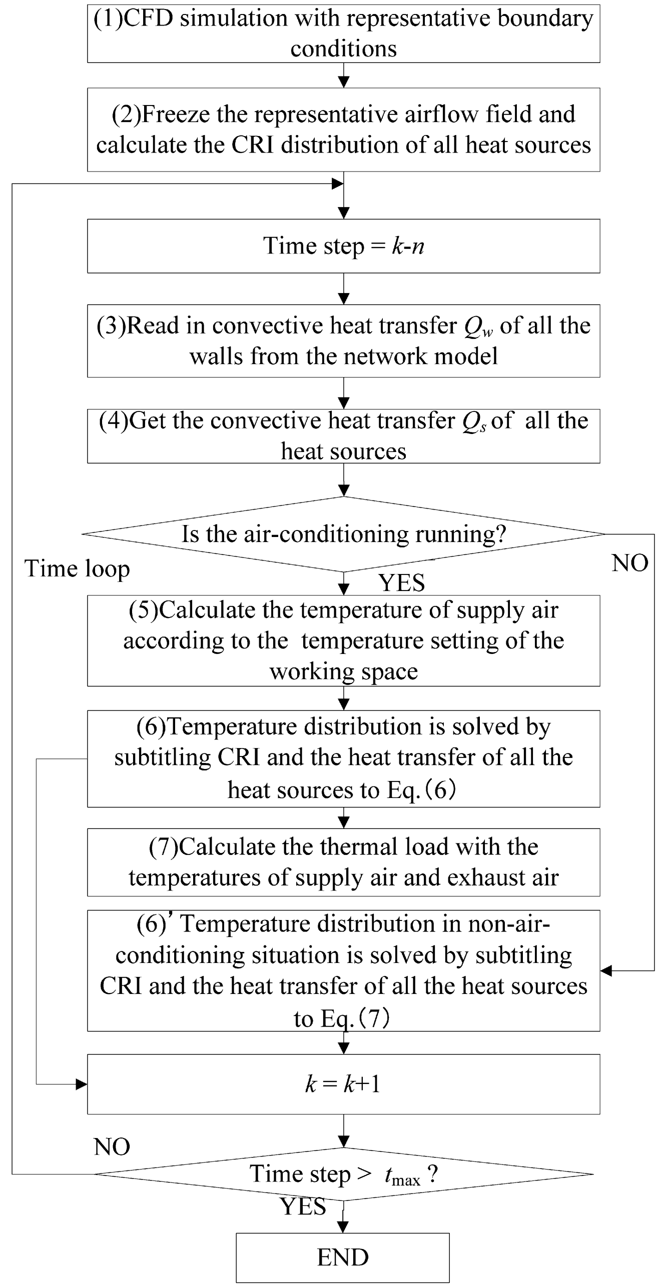

Fig. 5. Flow chart of proposed coupling simulation.

Then, by performing temperature calculations only for several times in the fixed representative airflow field, the CRI distribution of each heat source can be derived with Eqs. (1) and (2). After the preparatory CRI calculations are made, it is necessary to carry out

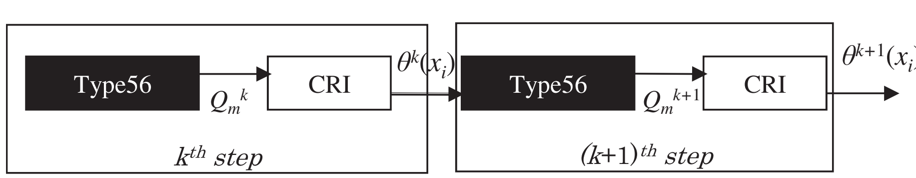

Fig. 4. Coupling of CRI and network model (Type56 of TRNSYS is used as the building energy simulation model).

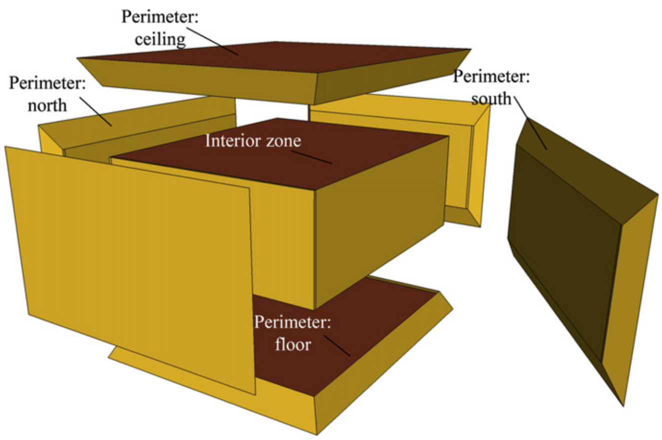

Fig. 6. Subdivision of space for calculation model.

an approach run for a couple of days in the energy simulation; this is represented as n steps.

In the CRI model, after reading in all of the convective heat transfers from various heat sources, the temperature distributions of the air-conditioned case and the non-air-conditioned case are calculated, respectively. Next, the detailed temperature distribution of the k step is averaged across the perimeter zones and the interior zone (Fig. 6), in which a conventional air-node zone in general energy solver is further divided into several perimeter zones and an interior zone. Each perimeter zone is composed of a wall and the vicinal air, having a relatively small width; this is used as the reference temperature for the heat transfer calculation at step $k + 1$. This process proceeds until the maximum time step is met.

In air-conditioning case, supply air temperature is determined by control strategy. For example, it can be a constant value; also it can be adjusted by the temperature of task area or living space at last time step. The involvement of temperature distribution to energy simulation makes this function available.

4. Results and analysis

4.1. Case study

A case study of a single office room with floor air-conditioning was carried out to demonstrate the proposed method. The size (6.0 m × 5.0 m × 3.0 m), number of openings (six floor inlets and four ceiling outlets) of the office room, and heat loads inside are illustrated in Fig. 7. The heat sources (and sinks) include three lighting fixtures, six heat-generating bodies, and heat transfer through construction materials. The convective heat from all of these heat factors is taken as heat sources, in accordance with the CRI concept. The air conditioning is set to work from 0800 to 1800 every day (an approximate workday schedule), and the heat load of the lighting fixtures and heat-generating bodies follows the same schedule. Except for the south wall, which is set as an external wall, all the other walls are set as joint walls connected to adjacent rooms having a constant temperature of 30 °C. The air-conditioning system is controlled to keep the temperature of the task area at 26 °C.

This study simulated two typical airflow fields. One is the air-conditioned case (during the working hours above), and the other is the non-air-conditioned case outside those hours. A summary of the numerical simulation conditions is listed in Table 1 and the airflow fields are illustrated in Figs. 8 and 9. In the air-conditioned case, the heat sources include the heat transfer from the six walls, the heat of lighting fixtures and heat-generating bodies, and the supply-air heat. In the non-air-conditioned case, only the heat transfer from the walls is taken as a heat source. In both airflow fields, the sub-temperature distribution was calculated for each heat source to get the CRI distribution.

4.2. Setting representative boundary conditions

Figs. 10 and 11 illustrate the CRI distribution of the heat transfer from the south wall in the air-conditioned and non-air-conditioned cases, respectively. The CRI distributions in the two cases are very different from each other, because the airflow fields are different.

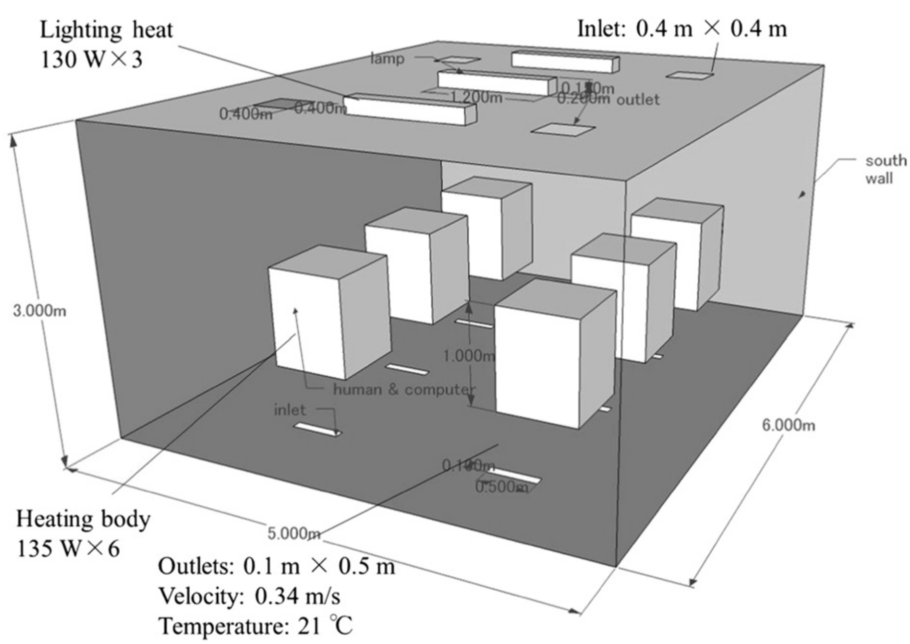

Fig. 7. Diagram of office room.

Table 1
Summary of numerical simulation.

Model	Standard $k-\varepsilon$ model
Mesh	136,670
Difference scheme	Temperature; velocity; advective term: secondary upwind; diffusion term: central difference
Boundary conditions	Velocity: generalized logarithmic law ($E = 9$) Heat transfer coefficient of external wall (south): 2.4 W/(m^2 K) Outdoor air temperature (daytime): 50 °C Outdoor air temperature (night): 26 °C Temperature of joint room: 30 °C Emissivity of inside walls: 0.8 Temperature: Generalized logarithmic law ($E = 9$)
Inlet and outlet	Outlet: Velocity = 0.34 m/s, Turbulence intensity = 0.1, Turbulence length scale = 0.1 m, Temperature of supply air = 21 °C Inlet: law of mass conservation, free slip Number of air changes: 4 times/h
CFD code	STARCCM+

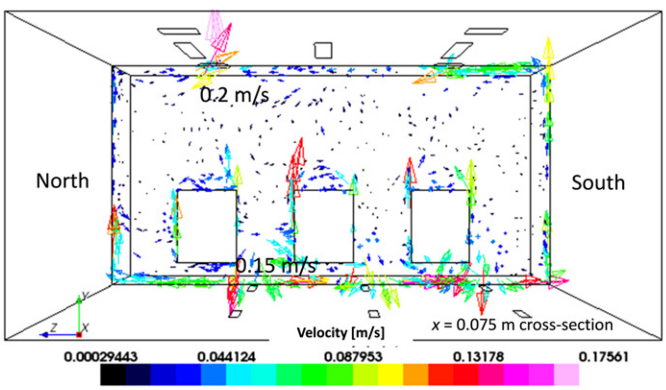

Fig. 8. Airflow field of air-conditioned case.

Thus, it is essential to decide the boundary conditions of the representative airflow field, which is affected by many factors.

In this study, only two airflow fields and thus only two sets of CRI values are included, because we focus only on demonstrating the coupling method. The average of the CRI distribution in each zone is calculated and listed in Tables 2 and 3.

One may question that for situations that the boundary conditions are different from the representative, such as office machines as on or off, temperature changes of walls and windows, and the impact of solar radiation, whether the setting of only two airflow fields is enough. Transient conditions do impact the local airflow

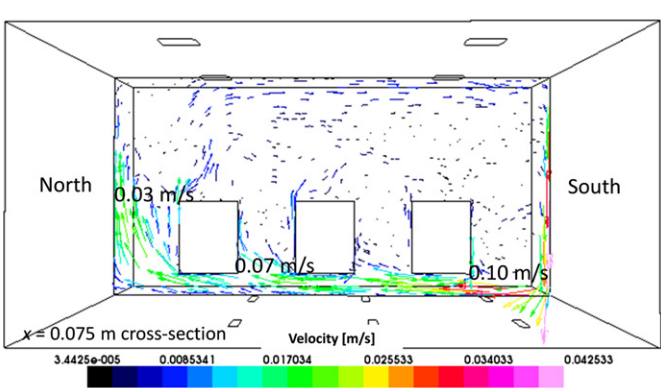

Fig. 9. Airflow field of non-air-conditioned case.

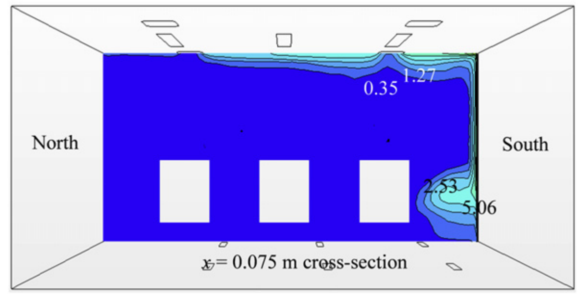

Fig. 10. CRI distribution of convective heat transfer of south wall as a heat source in air-conditioned case.

and further impact the local temperature distribution. In the real CFD simulation the calculation for temperature and velocity should be coupled. In the concept of CRI the impact of a heat source change to the airflow field is sacrificed. In another word, the coupled temperature and velocity calculations are separated. As a result, a fast temperature distribution calculation can be realized in a fixed airflow field.

In the case that machinery is shifted on or off, it would impact the volume of the upward airflow. We suggest setting the machinery as on when calculating the representative airflow field. And when it is off, according to Eqs. (6) and (7), it would have no impact to the temperature distribution. Wall and window temperature change would also bring the airflow change in the vicinity of the wall and window. Thus we suggest setting the wall temperature at an average condition when calculating the representative airflow field.

If these changes are not too large or are in the range of acceptance, few snapshots of CRI distribution are considered enough. Otherwise, more snapshots are needed to describe the reality. Then we have the question that which kind of boundary conditions should be taken to obtain the most representative airflow field for CRI calculation. The energy simulation is usually carried out at the design phase. During this stage the HVAC engineers would decide the positions of inlets and outlets, the expected controlled airflow field as well. The designed airflow field can be a good option for representative airflow field choosing.

The temperature distribution under specified condition can be seen as that under representative condition with a term indicting the sensitivity of the condition change. The CRI represents the sensitivity of the change of each heat source. The more sets of CRI under different conditions, the sensitivity of a heat source can be described more precisely. Theoretically, as the number of representative airflow fields increases, the result will approach that of a full CFD simulation and the computing cost will rise. As for the question of how many airflow fields would be necessary, it depends

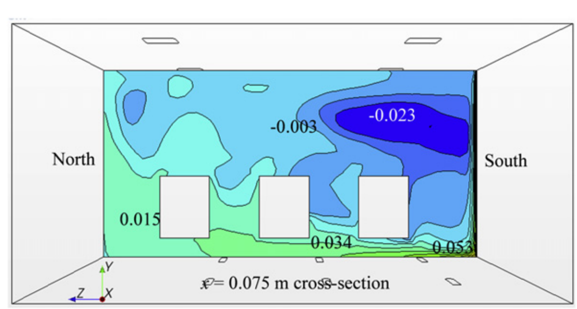

Fig. 11. CRI distribution of convective heat transfer of south wall as a heat source in non-air-conditioned case.

Table 2
CRIs of each zone in air-conditioned case.

Heat source	Zones							
	North	South	East	West	Ceiling	Floor	Interior	Task
North	3.52	0.29	0.80	0.96	1.71	0.20	0.94	0.59
South	0.20	3.67	0.71	0.89	2.77	0.42	0.60	0.50
East	1.01	0.95	3.50	0.05	1.90	0.26	0.76	0.42
West	0.86	0.75	0.06	3.72	2.01	0.24	0.88	0.54
Roof	1.54	0.25	1.58	1.69	5.27	0.00	0.31	0.02
Floor	1.17	0.90	1.02	1.03	1.00	0.81	1.05	0.95
Human	0.85	0.82	0.91	0.85	1.34	0.17	1.16	0.58
Light	1.12	0.07	0.73	0.71	4.22	0.00	0.15	0.01

CRI (contribution ratio of indoor climate) in air-conditioned case is defined as the ratio of the temperature rise (or decrease) caused by one individual heat source (or sink) at a location to the absolute value of the temperature rise (or decrease) caused by the same heat source with uniform distribution of the same amount of heat.

Table 3
CRIs of each zone in non-air-conditioned case.

Heat source	Zones							
	North	South	East	West	Ceil	Floor	Interior	Task
North	3.35	−0.78	−2.75	−2.02	1.85	−0.74	−0.01	−0.14
South	0.64	−0.15	−0.02	0.30	−0.26	1.17	−0.26	0.43
East	−1.11	−0.13	21.17	−14.54	2.43	−1.06	−0.94	−2.66
West	−1.26	0.46	−14.49	23.05	2.29	−0.42	−1.26	−1.00
Roof	−0.45	1.09	1.05	1.36	3.62	−0.10	−1.10	−1.09
Floor	1.34	−1.03	−0.46	−0.21	−0.18	0.58	−0.04	0.38

CRI (contribution ratio of indoor climate) in non-air-conditioned case is defined as the ratio of the temperature rise from the neutral temperature the heat released from heat source m.

on the precision demanded, and it also relates to the sensitivity of the CRI. Basically the more precise result is expected, the more snapshots of airflow field should be taken.

4.3. Results for 24 h

This study simulated an approach run for one week in August (Aug. 1–7) and the energy simulation for an additional week in August (Aug. 8–14) using a time step of 20 min with the standard weather data of Tokyo.

The results for the 24-h period on Aug. 14 are presented in Fig. 12 for the conditions where the task area temperature is set at 26 °C and the supply-air temperature is at about 18 °C. The temperature of the exhaust air is about 28 °C, which is higher than the set temperature. It can be seen from Fig. 13 that there is a large temperature range in the air-conditioned period. However, in the non-air-conditioned period, the indoor air is mixing well. Because the inlets are on the floor, the area near the floor has a much lower temperature than that of the other areas. The temperature of the ceiling area is much higher because of the lighting heat and buoyancy upstream. The temperature in the external south wall area fluctuated with the outdoor temperature. From the standpoint of thermal comfort, this low temperature may be not appropriate. However, our purpose here is to figure out the difference between models that consider air temperature and those that do not, so a large temperature distribution is necessary.

4.4. Comparison with non-coupled energy simulation

The results of CRI-coupled and non-coupled simulations under the same calculation conditions were compared. In the conventional energy simulation, heat is assumed to diffuse in the whole space uniformly and immediately. The temperature of the room or

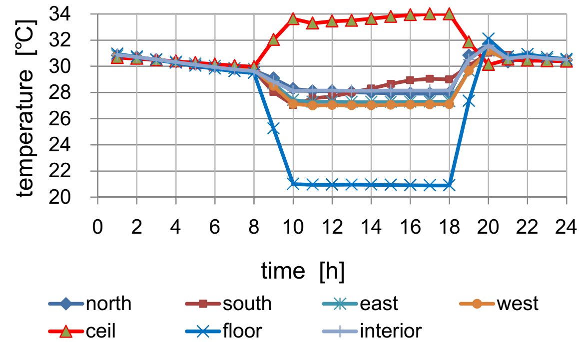

Fig. 13. Temperatures of all zones (8/14).

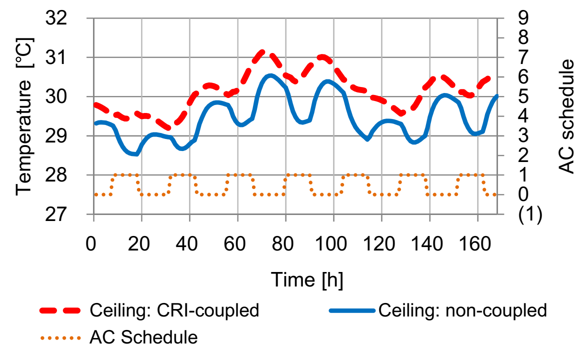

Fig. 14. Surface temperature of ceiling (8/8–8/14).

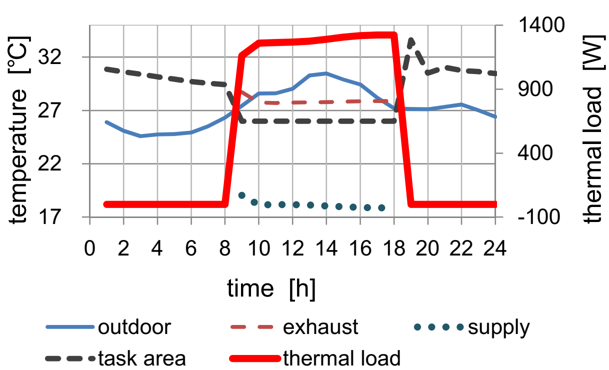

Fig. 12. Simulation results (8/14).

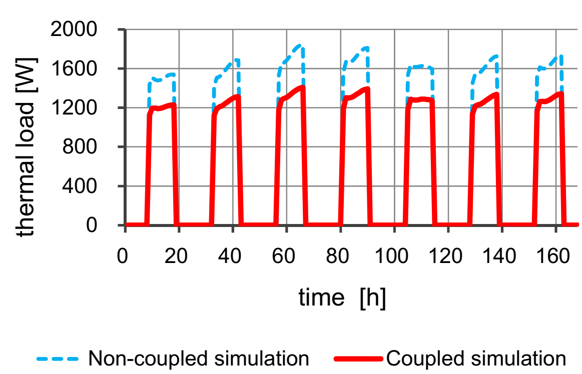

Fig. 15. Thermal load of coupled and non-coupled simulations (8/8–8/14).

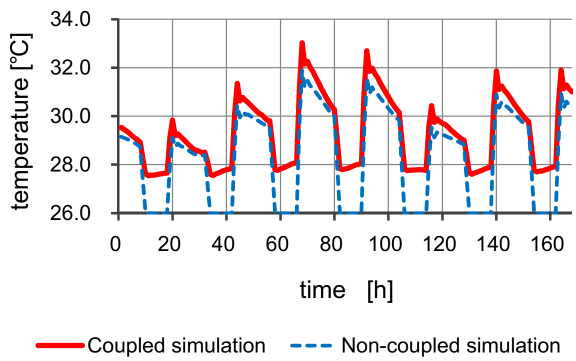

Fig. 16. Average temperature of coupled and non-coupled simulations (8/8–8/14).

zone is represented by the temperature of the air node. In the non-coupled simulation, the room temperature is kept uniformly at 26 °C, while as presented in Fig. 13, there is actually a large temperature distribution in air-conditioning duration. The temperatures in the perimeter zones are higher than 26 °C. Fig. 14 presents the surface temperatures of the walls, from which it can be seen that a relatively high temperature in the CRI-coupled simulation leads to lower heat transfer through construction materials. This can be seen from the results of the thermal load illustrated in Fig. 15. The thermal load of the air-conditioning system in the coupled simulation is 20% lower than in the non-coupled simulation. Fig. 16 presents the average temperatures of the two simulations. Because the temperature distribution is included and only the temperature of the task area needs to be kept at 26 °C in the CRI-coupled simulation, the average temperature of the whole space is about 28 °C, 2 °C higher than the set temperature.

5. Conclusions

This study has proposed a high-speed coupled program of building energy simulation and CRI (Contribution Ratio of Indoor climate), which is derived from CFD computation. This coupled method allows for a dynamic building energy simulation that considers room temperature distribution.

The introduction of CRI, which indicates the individual contribution to temperature distribution of each heat source in a fixed representative airflow field, can vastly reduce the computing cost, when compared to a directly coupled CFD simulation. In the CRI coupling method, a CRI model is used to calculate the temperature distribution based on the independent heat contributions of all of the heat sources to the temperature field. The general energy simulation is used to calculate the heat transfer of walls, with the air temperature near the wall as a reference temperature. This approach can increase the precision of the energy simulation, especially for the case that uses the temperature distribution positively to meet the needs of both thermal comfort and energy efficiency.

A case study of an office room with an under-floor air supply system was carried out to demonstrate the coupled program and test the sensitivity of the thermal load to temperature distribution. The energy demand result calculated by CRI-coupled method was 15–20% lower than that of a non-coupled network simulation. It also proved that the coupled simulation can produce more accurate and detailed results than could the non-coupled simulation.

For future studies, the sensitivity of the fixed airflow field needs to be investigated. First, when representative boundary conditions change, how much will the CRI calculated based on the representative airflow field affect the energy simulation results. Second, for different levels of required precision, the question of how many representative airflow fields are necessary needs to be investigated. The result calculated by CRI-coupled method will be compared to that calculated by CFD-coupled method to find out the error of this method.

Acknowledgments

The authors gratefully acknowledge the financial support of Grants-in-Aid for Scientific Research (No. 20234567 and No. 238444) from the Japan Society for the Promotion of Science (gs1).

References

[1] Hong T, Chou SK, Bong TY. Building simulation: an overview of developments and information sources. Building Environ 2000;35:347–61.
[2] EnergyPlus Documentation. Available from: http://apps1.eere.energy.gov/buildings/energyplus/energyplus_documentation.cfm.
[3] TRNSYS17. Available from: http://sel.me.wisc.edu/trnsys/features/features.html.
[4] Nielsen PV, Allard F, Awbi HB, Davidson L, Schälin A, editors. Computational fluid dynamics in ventilation design. REHVA: Federation of European Heating and Air-Conditioning Associations; 2007. REHVA guidebook No. 10.
[5] Bartak M, Beausoleil-Morrison I, Clarke JA, Denev J, Drkal F, Lain M, et al. Integrating CFD and building simulation. Building Environ 2002;37:865–71.
[6] Beausoleil-Morrison I. The adaptive coupling of heat and air modeling within dynamic whole-building simulation. Ph.D. thesis, University of Strathclyde, Glasgow, UK; 2000.
[7] Fan Y, Ito K. Energy consumption analysis intended for real office space with energy recovery ventilator by integrating BES and CFD approaches. Building Environ 2012;52:57–67.
[8] Zhai Z, Chen Q, Haves P, Klems JH. On approaches to couple energy simulation and computational fluid dynamics programs. Building Environ 2002;37:857–64.
[9] Zhai Z, Chen Q. Solution characters of iterative coupling between energy simulation and CFD programs. Energy Buildings 2003;37:493–505.
[10] Zhai Z, Chen Q. Performance of coupled building energy and CFD simulations. Energy Buildings 2005;37:333–44.
[11] Zhai Z, Chen Q. Sensitivity analysis and application guides for integrated building energy and CFD simulation. Energy Buildings 2006;38:1060–8.
[12] Sempey A, Inard C, Ghiaus C, Allery C. Fast simulation of temperature distribution in air-conditioned rooms by using proper orthogonal decomposition. Building Environ 2009;44:280–9.
[13] Kato S, Murakami S. New scales for assessing contribution of heat sources and sinks to temperature distributions in room by means of numerical simulation. Proc. ROOMVENT'94, fourth international conference on air distribution in rooms, Krakow, Poland, p. 539–57.
[14] Sasamoto T, Kato S, Zhang W. Control of indoor thermal environment based on concept of contribution ratio of indoor climate. Building Simulation 2010;3:263–78.
[15] Huang H, Kato S, Hu R, Ishida Y. Development of new indices to assess the contribution of moisture sources to indoor humidity and application to optimization design: proposal of $CRI_{(H)}$ and a transient simulation for the prediction of indoor humidity. Building Environ 2011;46:1817–26.
[16] Kato S, Murakami S. New ventilation efficiency scales based on spatial distribution of contaminant concentration aided by numerical simulation. ASHRAE Trans 1988;94:309–30.
[17] Matsumoto T, Kato S. Coupling CFD with energy simulation by CRI, ROOMVENT. 11th international conference on air distribution in rooms, Pusan, Korea; 2009. p. 1453–60.
[18] Hiyama K, Kato S. Integration of three-dimensional CFD results into energy simulations utilizing an advection–diffusion response factor. Energy Buildings 2011;43(I.10):2752–9.
[19] Hiyama K, Ishida Y, Kato S. Thermal simulation: response factor analysis using three-dimensional CFD in the simulation of air conditioning control. Building Simulation Int J 2010;3(3):195–203.
[20] Zhang W, Kato S, Ishida Y, Hiyama K. Calculation method of contribution ratio of indoor climate (CRI) by means of setting a uniform heat sink in natural convection air flow field. J Environ Eng (Trans AIJ);75:1027–32 [in Japanese].
[21] Zhang W, Kato S, Hiyama K, Ishida Y. Building energy simulation with considering air temperature distribution based on CFD. ROOMVENT 2011, 12th international conference on air distribution in rooms, Trondheim, Norway; 2011.