NRC Publications Archive Archives des publications du CNRC

Experimental evaluation of a numerical simulation model for predicting room air motion

Zhang, J. S.; Christianson, L. L.; Wu, G. J.; Zhang, R. H.

This publication could be one of several versions: author's original, accepted manuscript or the publisher's version. /
La version de cette publication peut être l'une des suivantes : la version prépublication de l'auteur, la version acceptée du manuscrit ou la version de l'éditeur.

Publisher's version / Version de l'éditeur:

Indoor Environment : The Journal of Indoor Air International, 2, 5-6, pp. 331-336, 1993-09

NRC Publications Record / Notice d'Archives des publications de CNRC:

<https://nrc-publications.canada.ca/eng/view/object/?id=a5411348-2598-4b3e-a612-d7768972b960>
<https://publications-cnrc.canada.ca/fra/voir/objet/?id=a5411348-2598-4b3e-a612-d7768972b960>

Access and use of this website and the material on it are subject to the Terms and Conditions set forth at

<https://nrc-publications.canada.ca/eng/copyright>

READ THESE TERMS AND CONDITIONS CAREFULLY BEFORE USING THIS WEBSITE.

L'accès à ce site Web et l'utilisation de son contenu sont assujettis aux conditions présentées dans le site

<https://publications-cnrc.canada.ca/fra/droits>

LISEZ CES CONDITIONS ATTENTIVEMENT AVANT D'UTILISER CE SITE WEB.

Questions? Contact the NRC Publications Archive team at

PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca. If you wish to email the authors directly, please see the first page of the publication for their contact information.

Vous avez des questions? Nous pouvons vous aider. Pour communiquer directement avec un auteur, consultez la première page de la revue dans laquelle son article a été publié afin de trouver ses coordonnées. Si vous n'arrivez pas à les repérer, communiquez avec nous à PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca.

<http://www.nrc-cnrc.gc.ca/irc>

Experimental evaluation of a numerical simulation model for predicting room air motion

NRCC-37041

Zhang, J.S.; Christianson, L.L.; Wu, G.J.;
Zhang, R.H.

September 1993

A version of this document is published in / Une version de ce document se trouve dans:
Indoor Environment : The Journal of Indoor Air International, 2, (5-6),
September 1993

The material in this document is covered by the provisions of the Copyright Act, by Canadian laws, policies, regulations and international agreements. Such provisions serve to identify the information source and, in specific instances, to prohibit reproduction of materials without written permission. For more information visit <http://laws.justice.gc.ca/en/showtdm/cs/C-42>

Les renseignements dans ce document sont protégés par la Loi sur le droit d'auteur, par les lois, les politiques et les règlements du Canada et des accords internationaux. Ces dispositions permettent d'identifier la source de l'information et, dans certains cas, d'interdire la copie de documents sans permission écrite. Pour obtenir de plus amples renseignements : <http://lois.justice.gc.ca/fr/showtdm/cs/C-42>

National Research
Council Canada

Conseil national
de recherches Canada

Canada

J.S. Zhang^a
L.L. Christianson^b
G.J. Wu^c
R.H. Zhang^b

^a Institute for Research in Construction,
National Research Council of Canada,
Ottawa, Ont., Canada;

^b Bioenvironmental Engineering Research
Laboratory, University of Illinois at
Urbana-Champaign, Ill.;

^c Thermal Storage Research Center,
University of Wisconsin, Madison, Wisc.,
USA

Experimental Evaluation of a Numerical Simulation Model for Predicting Room Air Motion

Key Words

Ventilation
Numerical simulation
Room air distribution

Abstract

Spatial distributions of air velocity, turbulent kinetic energy and temperature were measured in a full-scale office room ($7.32 \times 4.72 \times 2.44$ m) under non-isothermal conditions. Numerical simulation was also conducted with the computational fluid dynamics code 'EXACT3'. The numerical simulation results agreed with the measurements qualitatively but were quantitatively different. Possible reasons for these differences and future research needs are discussed in this paper.

Introduction

Predicting room air motion is important to the design and control of effective ventilation systems which can reliably provide satisfactory thermal comfort conditions and acceptable indoor air quality. With the advancement of computer technology and turbulence modelling, numerical simulation of room air and gas flow based on the computational fluid dynamics (CFD) technique has become a potential tool for designing ventilation systems. However, it is essential to validate numerical simulation models with reliable experimental data so that the simulation results can be used with confidence.

The validation task has been carried out most extensively by the International Energy Agency Annex 20¹ and

others [e.g. see ref. 2-6]. It is recognized that numerical simulation models must be validated for specific types of applications and ventilation conditions. The objective of this study was to measure detailed spatial distributions of air velocity, temperature and turbulent kinetic energy in a full-scale office room and compare the measured results with those obtained by a numerical simulation model.

Methods

Room Configuration and Test Conditions

Evaluating numerical simulation models for engineering design purposes requires well-defined test cases which represent realistic and typical ventilation conditions and room configurations. A full-scale office room with a realistic airflow rate, internal heat load and furniture arrangement was chosen for this purpose (fig. 1). In this paper, results are presented for a non-isothermal test condition with an empty room. The test conditions were (see Nomenclature for definitions of variables): $U_o = 3.05$ m/s, $T_d = 12.8$ °C, $T_e = 23.3$ °C, $T_f = 40.0$ °C, $\Delta T_{ed} = 16.7$ °C, $Re_d = 2,016$, $Ar_{ed} = 3.64 \times 10^{-4}$ and $Q = 0.21$ m³/s.

¹ Annex 20 is a cooperative research program sponsored by the International Energy Agency with the participation of experts of room air motion and building environment studies from 10 European and North American countries, including the USA and Canada.

Fig. 1. Schematic of the test room. 1 = Book shelves; 2 = desks; 3 = partitions.

Experimental Facility and Procedure

Experiments were conducted with a room ventilation simulator developed for room air and gas distribution studies [7, 8]. The dimensions of the full-scale office (fig. 1) were $7.32 \times 4.72 \times 2.44$ m. The walls, ceiling and floor of the room were well-insulated with R values of 2.11, 2.11 and $3.35 \text{ }^\circ\text{C} \cdot \text{m}^2/\text{W}$, respectively. Air for the test room was supplied through a continuous diffuser slot with a discharge angle of 25° from the ceiling. The diffuser air temperature was kept constant by an independent cooling unit. The internal heat loads were simulated by 41 heating panels (0.61×1.22 m) uniformly distributed over the floor surface. During the experiments, temperatures at the diffuser, exhaust, centre of the room and the floor surface were monitored by thermocouples and recorded with a data logger to ensure stable experimental conditions. The air temperature around the test room was maintained within $\pm 1.5^\circ\text{C}$ of the temperature at the centre of the test room by a separate air-conditioning system, so that the heat transfer through the walls and ceiling were minimized (i.e. the adiabatic conditions at the surfaces of the walls and ceiling were well-approximated).

Room airflow patterns were visualized with neutrally thermal buoyant helium bubbles. Air velocities and temperatures inside the test room were measured with a hot-wire anemometer and resistance temperature detectors (RTD), respectively. A microcomputer-based automatic data acquisition and probe-positioning system [8] was used to collect data. The flow within the room was practically two-dimensional [9]. Measurements were taken at 22×41 grid points at the symmetric plane of the room to provide detailed data (fig. 2a). The uncertainties (ϵ) involved with the experiments were as follows [8]:

$$\begin{aligned} \epsilon_{U_d} &\approx \pm 1.6\% \text{ or } 0.015 \text{ m/s, whichever is larger;} \\ \epsilon_{T_d} &\approx \pm 1^\circ\text{C;} \\ \epsilon_{U_r} &\approx \pm 25, \pm 12, \pm 8, \pm 5 \text{ and } \pm 4\% \text{ for velocities of } 0.05\text{--}0.10, \\ &\quad 0.10\text{--}0.15, 0.15\text{--}0.25, 0.25\text{--}0.51 \text{ and } >0.51 \text{ m/s, respectively;} \end{aligned}$$

and

$$\epsilon_T \approx \pm 0.4^\circ\text{C,}$$

where, ϵ_{U_d} , ϵ_{T_d} , ϵ_U and ϵ_T are uncertainties for diffuser air velocity, diffuser air temperature, room air velocities and room air temperatures, respectively. These uncertainties are important for interpreting the experimental results and for comparing measurements with numerical simulations.

Numerical Simulation Procedure

The computer code 'EXACT3' [10] was used for numerical simulation of the air and gas distribution within the office room. In 'EXACT3', the averaged three-dimensional Navier-Stokes equations coupled with the high Reynolds number k - ϵ turbulence model are solved numerically using the Marker and Cell finite difference method.

Beginning with the proper specifications of air conditions at the diffuser is critical to the success of numerical simulations of room airflow since they have the greatest influence on the room airflow pattern. Ideally, measured profiles of air velocity, temperature and turbulent kinetic energy at the diffuser are preferred for numerical simulations that are compatible with the experiments [5]. However, in most realistic cases, it is practically difficult (if not impossible) to measure the profiles of air velocity, temperature and turbulent kinetic energy, and these data are usually not available during the design stage. Therefore, certain types of approximations are usually necessary in specifying the boundary conditions at the diffuser.

For the simulation results presented in this paper, uniform profiles of air velocity, temperature and turbulent kinetic energy were assumed at the diffuser, but the magnitude of the diffuser air velocity was specified to provide an equal amount of jet momentum measured in the fully developed turbulent region of the diffuser air jets. Experiments showed that the jet momentum had a more significant effect on the room air motion than did the airflow rate [8]. Specifying diffuser air velocity based on the jet momentum has also been found to be important in previous numerical simulations [11].

The turbulent kinetic energy and temperature at the diffuser were specified based on the measurement at the centre of the diffuser slot, while the dissipation rate of turbulent kinetic energy was estimated by the following equation:

$$\epsilon = C_\mu^{0.75} \frac{k^{1.5}}{l} \quad (1)$$

where ϵ = dissipation rate of turbulent kinetic energy, $C_\mu = 0.09$, k = turbulent kinetic energy and $l = 0.03$ times the hydraulic diameter of the opening slot.

Surfaces of the ceiling and walls were assumed to be adiabatic. A uniform heat flux was specified for the floor surface.

The simulation results presented in this paper were obtained with a $50 \times 44 \times 23$ non-uniform grid scheme. Simulation with a finer grid scheme ($60 \times 50 \times 23$) did not produce a different result in terms of room airflow patterns, so the $50 \times 44 \times 23$ grid density was considered to be appropriate.

Simulations were conducted with a workstation computer which has 28.5 MIPS and 4.2 MFLOPS of integer and floating point performance, respectively. It took about 7 days CPU time to obtain the steady-state solution. The solution was considered to be steady state based on the following criteria: (1) the distribution patterns of air velocity and temperature remained unchanged; (2) the volumetric room mean square residuals of the equations of momentum, temperature, turbulent kinetic energy and dissipation rate of turbulent kinetic energy were less than 5×10^{-5} .

a Visualized (measurement grid is also shown)

b Computed → 0.25 m/s

Fig. 2. Comparison between visualized and computed airflow patterns at the symmetric plane.

Results

Airflow Patterns

The observed airflow pattern showed an inclined diffuser air jet that attached to the ceiling due to the well-known Coanda effect and remained attached to the ceiling for a certain distance before it separated from the ceiling (fig. 2a). A reverse flow was formed under the diffuser air jet due to the entrainment of the jet. It can be seen that part of the jet fell to the occupied zone (defined as the region from the floor to 1.83 m and 0.30 m from each wall before it reached the opposite wall).

The computed airflow pattern (fig. 2b) agreed generally well with the observed pattern, but the jet did not drop until it reached the opposite wall, and the entire occupied zone was ventilated by the reverse flow. The computed jet also had a narrower spread than that observed with the helium bubbles.

a Measured

b Computed

Fig. 3. Comparison between measured and computed contour maps of air velocity (m/s) at the symmetric plane.

Air Velocity Distribution

The contour map of the measured velocity distribution shows a decayed air jet, as expected (fig. 3a). If the 0.25 m/s contour line is regarded as the jet boundary, one can see the jet spread to the occupied zone before it reached the opposite wall, which was in agreement with the airflow pattern observed. Velocities in the occupied zone were fairly uniform (0.10–0.25 m/s) with low values at the centre of the large recirculation eddy (fig. 2a), as expected.

The computed velocity distribution pattern (fig. 3b) was similar to the measurement in general. That is, it predicted high velocities in the regions close to the ceiling, walls and floor and low velocities in the central region of the room. However, the computed contour map shows a slower decay of jet velocity, which results in higher velocities in the region close to the ceiling, walls and floor as compared with the measurements. The numerical model did predict low-velocity levels (0.10–0.20 m/s) at the central region of the room, which were similar to the measured values.

Fig. 4. Comparison between measured and computed contour maps of turbulent kinetic energy [$100 \times (\text{m/s})^2$] at the symmetric plane.

Turbulent Kinetic Energy Distribution

Measurements showed high turbulent kinetic energy in the jet region and low values in the occupied region, as expected (fig. 4a). There was relatively high turbulent kinetic energy close to the floor surface as compared with that at the centre of the occupied zone, which was due to the heat generation and the relatively higher velocity gradient over the surface.

The computed turbulent kinetic energy was significantly higher than the measurements (fig. 4b). This was due to the underestimation of the jet decay in the numerical model, since a higher velocity in the jet would cause a higher velocity gradient and cause more turbulence. The higher air velocity over the surfaces of the opposite wall and the floor would also cause more turbulence to be produced over these surfaces as compared with that in the experiment.

Spatial Distribution of Temperature

The contour map (fig. 5a) of the measured temperatures indicates that the incoming air temperature increased as the air travelled and mixed with the room air.

Fig. 5. Comparison between measured and computed contour maps of temperature ($^{\circ}\text{C}$) at the symmetric plane.

Relatively high temperatures were present close to the floor surface due to the heat production there.

The numerical simulation predicted similar temperature distributions to the measured pattern (fig. 5b), but the predicted temperatures were slightly lower than the measurements in the occupied region.

Discussion

According to the above results, the numerical predictions are in reasonable agreement with the experimental measurements in terms of airflow pattern and the distribution patterns of air velocity, temperature and turbulent kinetic energy. However, the numerical simulation predicted slower jet decay, narrower jet spread and a delayed drop of the jet as compared with the measurements. As a result, the simulation overestimated the air velocity in the regions adjacent to the surfaces of the ceiling, walls and floor, and the turbulent kinetic energy in the room. The temperature in the room was underestimated.

These differences between the prediction and the measurement appear to indicate that the thermal buoyancy effect was not sufficiently accounted for in the numerical model, since the thermal buoyancy would speed up the jet decay, increase the jet spread and cause an early drop of the jet. However, the differences may be also due to the small three-dimensional effect present in the experiment. There can be up to 3% nonuniformity in the distribution of diffuser air velocity along the length of the diffuser slot (i.e. in direction z) and in the heat generation on the floor. This non-uniformity would cause air motion in the third direction (direction z). Such a three-dimensional effect would reduce the Coanda effect and result in an earlier drop of the jet. There were also uncertainties involved in the measurements, as described earlier, which might have added to the differences between the numerical and experimental results. For example, the single hot-wire probe used in the present measurement was not sensitive to the velocity in the third direction, which would cause error in the velocity measurement if there were a velocity component in that direction. ASHRAE [12] is currently sponsoring a research project to measure three-dimensional room air velocities for validating numerical simulation models.

The results showed that the predicted temperatures in the occupied region were lower than the measurements. This may be partially due to the radiation effect which was not modelled in the simulation code. Radiant heat exchange within the room, which obviously existed in the experiment, would make the room air temperature more uniform and result in a higher temperature in the occupied region when compared with the numerical simulation.

The differences between the simulation and measurement were also partially due to the approximation in specifying the boundary conditions in the simulation. In the practical application of numerical simulation, approximations in specifying the boundary conditions are unavoidable. Studies are needed to quantify the uncertainty involved in the simulation results due to this approximation in the boundary conditions.

Air distribution in a real room is a very complicated process. A systematic evaluation of the existing numerical simulation models should start with the simplest case (simple geometry, no internal heat load and no internal obstruction), and then gradually added complexity (internal heat load, obstructions and more complicated geometry). In this way, one can determine when the numerical models start to break down. Guidelines for the application of the models can be established once the limitations of the models are identified.

Conclusions

Systematic evaluation of numerical models of room air motion should be conducted from simple to complicated cases so that the limitations of the models can be clearly identified. Guidelines for using the numerical simulation technique can then be developed. For engineering design purposes, essential are the quantification of the uncertainties that may be involved in the specification of the boundary conditions of numerical simulations and the assessment of how such uncertainties affect simulation results. Criteria are also needed to judge what prediction accuracy is sufficient for engineering design purposes.

Nomenclature

- Ar_{fd} = Archimedes number defined as $\frac{\beta g w_d (T_f - T_d)}{U_d^2}$;
- $E(f)$ = spectral density function of velocity fluctuations, $(m/s)^2$;
- f = frequency, Hz;
- g = gravitational acceleration rate, m/s^2 ;
- H = room height, m;
- h = convective heat transfer coefficient, $W/m^2 \cdot K$;
- k = turbulent kinetic energy, $(m/s)^2$;
- L = length of the room (in direction z), m;
- l_d = length of the diffuser slot (in direction z), m;
- Q = ventilation rate, m^3/s ;
- q''_f = heat flux from the floor surface, W/m^2 ;
- Re = Reynolds number defined as $\frac{U_d W_d}{\nu}$;
- T_f = maximum temperature in room (e.g. on the heated surface), $^{\circ}C$;
- T_d = diffuser air temperature, $^{\circ}C$;
- T_e = air temperature at the exhaust, $^{\circ}C$;
- ΔT_{ed} = $T_e - T_d$, $^{\circ}C$;
- u' = standard deviation of velocity, m/s;
- U_o = nominal average air velocity at the diffuser calculated from the measured ventilation rate, m/s;
- U_d = average diffuser air velocity calculated from the measured jet momentum, m/s;
- U_e = average air velocity at the exhaust based on the mass balance, m/s;
- W = width of the test room (in direction x), m;
- w_d = width of the diffuser slot (in direction y), m;
- w_e = slot width of the exhaust, m;
- x, y, z = Eulerian Cartesian coordinates, m;
- y_d = y coordinate of the diffuser left edge, m;
- y_e = y coordinate of the exhaust right edge, m;
- β = thermal expansion coefficient, $1/K$;
- ν = kinematic viscosity, m^2/s ;
- ρ = density of diffuser air, kg/m^3 ; and
- ϵ_e = dissipation rate of the turbulent kinetic energy at the diffuser, $(m/s)^2/s$.

References

- 1 Whittle GE, Clancy EM: Evaluation of cases B, D, E - Presentation of results from measurements and simulations of test cases B, D, and E. IEA Annex 20 Research Report No. 1.22. ETH Zentrum, Switzerland, International Energy Agency, 1991.
- 2 Murakami S, Kato S, Kondo Y: Numerical prediction of horizontal non-isothermal 3-D jet in room based on algebraic second-moment closure model. ASHRAE Trans 1992;98:951-962.
- 3 Launder BE: Second-moment closure: Present ... and future? Int J Heat Fluid Flow 1989; 10:282-300.
- 4 Chen Q, Jiang Z: Significant questions in predicting room air motion. ASHRAE Trans 1992;98:929-939.
- 5 Zhang JS, Wu GJ, Christianson LL: Detailed measurements of room air distribution for evaluating numerical simulation models. ASHRAE Trans 1992;98:58-65.
- 6 Kuehn TH, Pui DYH, Gratzek JP: Experimental and numerical results for air flow and particle transport in a microelectronics clean room. ASHRAE Trans 1992;98:940-950.
- 7 Wu GJ, Christianson LL, Zhang JS, Riskowski GL: Adjustable room ventilation simulator for room air and air contaminant distribution modelling. Indoor Air '90, Proc 5th Int Conf on Indoor Air Quality and Climate, Toronto, July 29-August 3, 1990, vol 4, pp 237-242.
- 8 Zhang JS: A Fundamental Study of Two-Dimensional Room Ventilation Flows under Isothermal and Non-Isothermal Conditions; PhD thesis University of Illinois at Urbana-Champaign, 1991.
- 9 Wu GJ: Study of Jet Characteristics for Cold Air Distribution; PhD thesis University of Illinois at Urbana-Champaign, 1992.
- 10 Kurabuchi T, Fang JB, Grot RA: A Numerical Method for Calculating Indoor Airflows Using a Turbulence Model. Research Report NISTIR 89-4211. Gaithersburg, National Institute of Standards and Technology, 1990.
- 11 Chen Q, Moser A: Simulation of a multiple-nozzle diffuser; in Proc 12th AIVC Conference, Ottawa, September 1991. Coventry (UK), Air Infiltration and Ventilation Centre, vol 2, pp 1-14.
- 12 ASHRAE: Room Air Movement Data for Validating Numerical Models. ASHRAE Research Work Statement No. 703-TRP. Atlanta, American Society of Heating, Refrigeration and Air-Conditioning Engineers, 1991.