#6258

AN EXPERIMENTAL EVALUATION OF A NUMERICAL SIMULATION MODEL FOR PREDICTING ROOM AIR MOTION

J.S. Zhang
Institute for Research in Construction
National Research Council of Canada
Ottawa, Canada K1A 0R6

L.L. Christianson
Bioenvironmental Engineering Research Laboratory
University of Illinois at Urbana-Champaign
Urbana, Illinois 61801

G.J. Wu
Thermal Storage Research Center
University of Wisconsin
Madison, Wisconsin

R.H. Zhang
Bioenvironmental Engineering Research Laboratory
University of Illinois at Urbana-Champaign
Urbana, Illinois 61801

Spatial distributions of air velocity, turbulent kinetic energy and temperature were measured in a full scale office room (24x15.5x8 ft [7.32x4.72x2.44 m]) under non-isothermal conditions. Numerical simulation was also conducted with the CFD code "EXACT3". The numerical simulation results agreed with the measurements qualitatively, but were quantitatively different from the measurements. Possible reasons for these differences and future research needs are discussed in the 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 the 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 20[1] and others (e.g., see reference [2] through [6]). It is recognized that numerical simulation models must be validated for specific type 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

[1] Annex 20 is an International Energy Agency sponsored cooperative research program with participation of experts of room air motion and building environment studies from U.S., Canada and 10 European and North American countries including U.S.A. and Canada.

a full scale office room, and compare the measured results with those obtained by a numerical simulation model. Preliminary results are presented in this paper.

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 realistic airflow rate, internal heat load and furniture arrangement was chosen for this purpose (Figure 1). In this paper, results are presented for a non-isothermal test condition with no furniture within the room (Table 1).

TABLE 1 Test Conditions*

U_o (ft/min)	T_d (F)	T_e (F)	T_f (F)	ΔT_{ed} (F)	Re_d	Ar_{ed} ($x10^4$)	Q (ft^3/min)
600	55.0	74	104	19.0	2016	3.64	450

* The room was empty in this test case.

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 (Figure 1) were 24x15.5x8 ft (7.32x4.72x2.44 m). The walls, ceiling and floor of the room were well insulated with R-Values of 12, 12 and 19 F·ft²·hr/Btu (2.11, 2.11, and 3.35 RSI), respectively. Air for the test room was supplied through a continuous diffuser slot with an discharge angle of 25 degree 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 (2 by 4 ft [0.61 by 1.22 m]) uniformly distributed over the floor surface. During experiments, temperatures at the diffuser, exhaust, centre of the room and the floor surface were monitored by thermocouple and recorded with a data logger to ensure stable experimental conditions. The air temperature around the test room was maintained within ±3 F (±1.5 °C) of the temperature at the centre of the test room by an separate air conditioning system, so that the heat transfer through walls and ceiling were minimized (i.e., the adiabatic conditions at the surfaces of 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 RTD temperature sensors, 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 22x41 grid points at the symmetric plane of the room to provide detailed data (Figure 2). The uncertainties (e) involved with the experiments were as follows [8]:

$e_{Ud} \approx \pm 1.6\%$ or ±3 ft/min, which ever is larger

$e_{Td} \approx \pm 1.8$ F

$e_U \approx \pm 25\%, \pm 12\%, \pm 8\%, \pm 5\%$ and ±4% for velocities of 10 to 20 ft/min, 20 to 30 ft/min, 30 to 50 ft/min, 50 to 100 ft/min and >100 ft/min, respectively.

$e_T \approx \pm 0.7$ F

where, e_{Ud}, e_{Td} and e_U 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 (MAC) finite difference method.

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 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 effects on room air motion than the airflow rate [8]. Specifying diffuser air velocity based on the jet momentum was also 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:

$$\varepsilon = C_\mu^{0.75} \frac{k^{1.5}}{l} \tag{1}$$

where,

ε = dissipation rate of turbulent kinetic energy;

C_μ = 0.09;

k = turbulent kinetic energy;

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 was obtained with a 50x44x23 non-uniform grid scheme. Simulation with a finer grid scheme (60x50x23) did not produce a different result in terms of room airflow patterns, so the 50x44x23 grid density was considered to be appropriate.

Simulations were conducted with a work station computer which has 28.5 MIPS and 4.2 MFLOPS of integer and floating point performance, respectively. It took about 7 day 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 equations of momentum, temperature, turbulent kinetic energy and dissipation rate of turbulent kinetic energy were less than $5x10^{-5}$.

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 (Figure 2). 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 floor to 6 ft high and 1 ft from each walls) before it reached the opposite wall.

The computed airflow pattern (Figure 3) 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.

Air Velocity Distribution

The contour map of the measured velocity distribution shows a decayed air jet as expected (Figure 4). If the 50 ft/min (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 are fairly uniform (20 to 50 ft/min [0.10 to 0.25 m/s]) with low values at the centre of the large recirculation eddy (Figure 2) as expected.

The computed velocity distribution pattern (Figure 5) is similar to the measurement in general. That is, it predicted high velocities in the regions close to ceiling, walls and the floor and low velocities in the central region of the room. However, the computed contour map shows an slower decay of jet velocity, which results in higher velocities in the region close to the ceiling, walls and floor as compared to the measurements. The numerical model did predict low velocity levels (20 to 40 ft/min) at the central region of the room, which were similar to the measured values.

Turbulent Kinetic Energy Distribution

Measurements showed high turbulent kinetic energy in the jet region and low in the occupied region as expected (Figure 6). There was relatively high turbulent kinetic energy close to the floor surface as compared to 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. This was due to the under estimation 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 production over these surfaces as compared to the experiment.

Spatial Distribution of Temperature

The contour map (Figure 8) of the measured temperatures indicates that the incoming air temperature increased as the air travelled and mixed with the room air. Relatively high temperatures were present close to the floor surface due to the heat production there.

The numerical simulation predicted similar to the measured pattern of temperature distributions (Figure 9), but slightly under predicted the temperature in the occupied region.

DISCUSSIONS

According to the above results, the numerical predictions have a reasonable agreement with the experimental measurement in terms of airflow pattern and 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 to the measurements. As a result, the simulation over predicted the air velocity in the regions adjacent to the surfaces of ceiling, walls and floor, and the turbulent kinetic energy in the room. The temperature in the room was under predicted.

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% non-uniformity in the distribution of diffuser air velocity along the length of the diffuser slot (i.e., in z direction) and in the heat generation on the floor. These non-uniformity would cause air motion in the third direction (i.e., z direction). Such three dimensional effect would reduce the Coanda effect and result in earlier drop of the jet. There were also uncertainties involved in the measurements as described earlier, which might 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 was velocity component in the z 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 was 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 compared to 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 practical application of numerical simulation, approximations in the specifying boundary conditions are unavoidable. Studies are needed to quantify the uncertainty involved in the simulation results due to the approximation in the boundary conditions.

Air distribution in a realistic 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 add complexity (internal heat load, obstructions and more complicated geometry) to it. 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.

SUMMARY AND CONCLUSIONS

Velocity, turbulent kinetic energy and temperature were measured in detail at the symmetric plane of the test room in which flow was weakly thermal buoyant. The corresponding numerical simulation results agreed with the measurements qualitatively, but were quantitatively different from the measurements. These differences appear to be due to the insufficient account of the thermal buoyancy effect in the numerical simulation model, but may also be due to the difficulty in duplicating the experimental conditions and the uncertainties involved in the experiments and in the specification of boundary conditions. Systematic evaluation of numerical models of room air motion should be conducted from simple to complicated cases so the limitations of models can be clearly identified. Guidelines for using the numerical simulation technique can then be developed.

For engineering design purpose, it is also essential to quantify the uncertainties that may be involved in the specification of the boundary conditions of numerical simulations and how will such uncertainties be affect to the simulation results. Criteria are also needed to judge what prediction accuracy is sufficient for engineering design purpose.

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, $(ft/min)^2$ or $(m/s)^2$;
f	= Frequency, Hz;
g	= Gravitational acceleration rate, ft/min^2 (m/s^2);
H	= Room height, ft (m);
h	= Convective heat transfer coefficient, $Btu/(h \cdot ft^2 \cdot F)$ ($W/m^2 \cdot K$);
k	= Turbulent kinetic energy, $(ft/min)^2$ or $(m/s)^2$;
L	= Length of the room (in Z direction), ft (m);
l_d	= Length of the diffuser slot (in Z direction), ft (m);
Q	= Ventilation rate, ft^3/min (m^3/s);
q''_f	= Heat flux from the floor surface, Btu/ft^2h (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), F (°C);
T_d	= Diffuser air temperature F (°C);
T_e	= Air temperature at the exhaust, F (°C);
ΔT_{ed}	= T_e-T_d, F (°C);
u'	= Standard deviation of velocity, ft/min (m/s);
U_o	= Nominal average air velocity at the diffuser calculated from the measured ventilation rate, ft/min;
U_d	= Average diffuser air velocity calculated from the measured jet momentum, ft/min;
U_e	= Average air velocity at the exhaust based on the mass balance, ft/min;
W	= Width of the test room (in X direction), ft (m);
w_d	= Width of the diffuser slot (in Y direction), ft (m);
w_e	= Slot width of the exhaust, ft (m);
x, y, z	= Eulerian Cartesian coordinates, ft (m);
y_d	= y coordinate of the diffuser left edge, ft (m);
y_e	= y coordinate of the exhaust right edge, ft (m);
β	= Thermal expansion coefficient, 1/R (1/K);
ν	= Kinematic viscosity, ft^2/min (m^2/s);

ρ = Density of diffuser air, lb_m/ft^3 (kg/m^3);
ε_c = Dissipation rate of the turbulent kinetic energy at the diffuser, ft^2/min^3;

REFERENCES

[1] Whittle, G.E. and E.M. Clancy. 1991. "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.

[2] Murakami, S., S. Kato and Y. Kondo. 1992. "Numerical prediction of horizontal non-isothermal 3-D jet in room based on algebraic second-moment closure model". ASHRAE Transactions 98(1).

[3] Launder, B.E. 1989. "Second-moment closure: present... and future ?". Int. J. Heat and Fluid Flow, Vol. 10, pp. 282-300.

[4] Chen, Q. and Z. Jiang. 1992. "Significant questions in predicting room air motion". ASHRAE Transactions 98(1).

[5] Zhang, J.S., G.J. Wu and L.L. Christianson. 1992. Detailed measurements of room air distribution for evaluating numerical simulation models. ASHRAE Transactions 98(1).

[6] Kuehn, T.H., D.Y.H. Pui and J.P. Gratzek. 1992. "Experimental and numerical results for air flow and particle transport in a microelectronics clean room". ASHRAE Transactions 98(1).

[7] Wu, G.J., L.L. Christianson, J.S. Zhang and G.L. Riskowski. 1990. Adjustable room ventilation simulator for room air and air contaminant distribution modelling. In Indoor Air '90, Proc. 5th international conference on indoor air quality and climate, 4:237-242.

[8] Zhang, J.S. 1991. An Fundamental Study of Two Dimensional Room Ventilation Flows under Isothermal and Non-isothermal Conditions. PhD Thesis, University of Illinois at Urbana-Champaign, Urbana, IL.

[9] Wu, G.J. 1992. Study of Jet Characteristics for Cold Air Distribution. PhD Thesis, University of Illinois at Urbana-Champaign, Urbana, IL.

[10] Kurabuchi, T., J.B. Fang and R.A. Grot. 1990. A numerical method for calculating indoor airflows using a turbulence model. National Institute of Standards and Technology, Research Report NISTIR 89-4211.

[11] Chen, Q., and A. Moser. 1991. Numerical simulation of a complicated diffuser air jet. Proc. 12th AIVC Conference. Ottawa, Canada.

[12] ASHRAE. 1991. Room air movement data for validating numerical models. ASHRAE Research Workstatement #703-TRP.

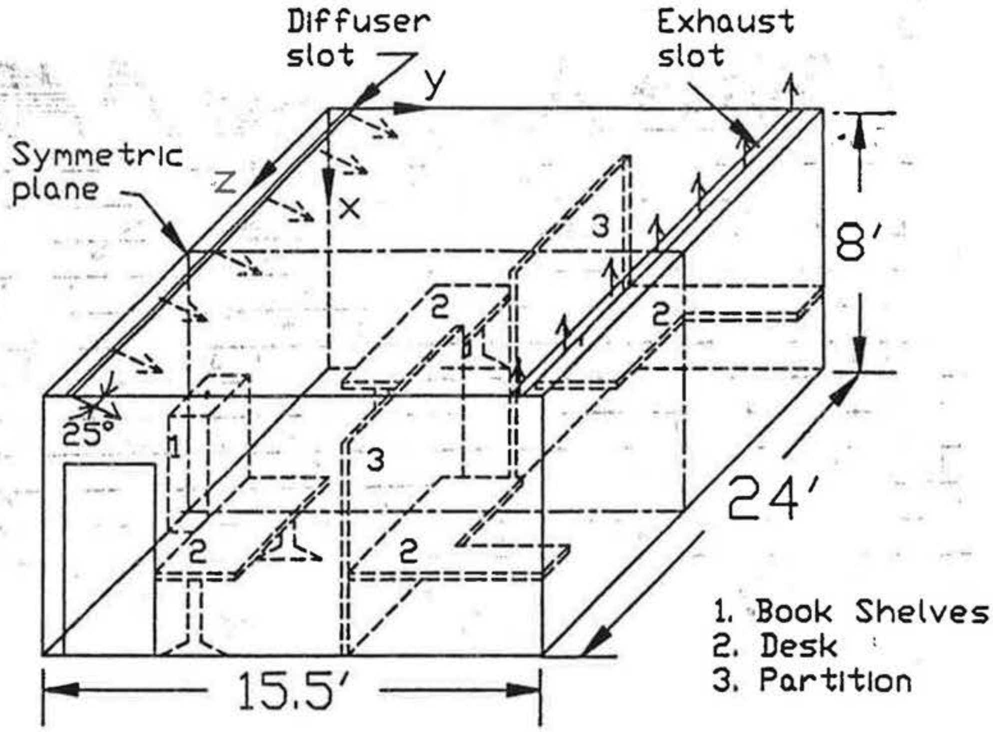

Figure 1 Schematic of the test room

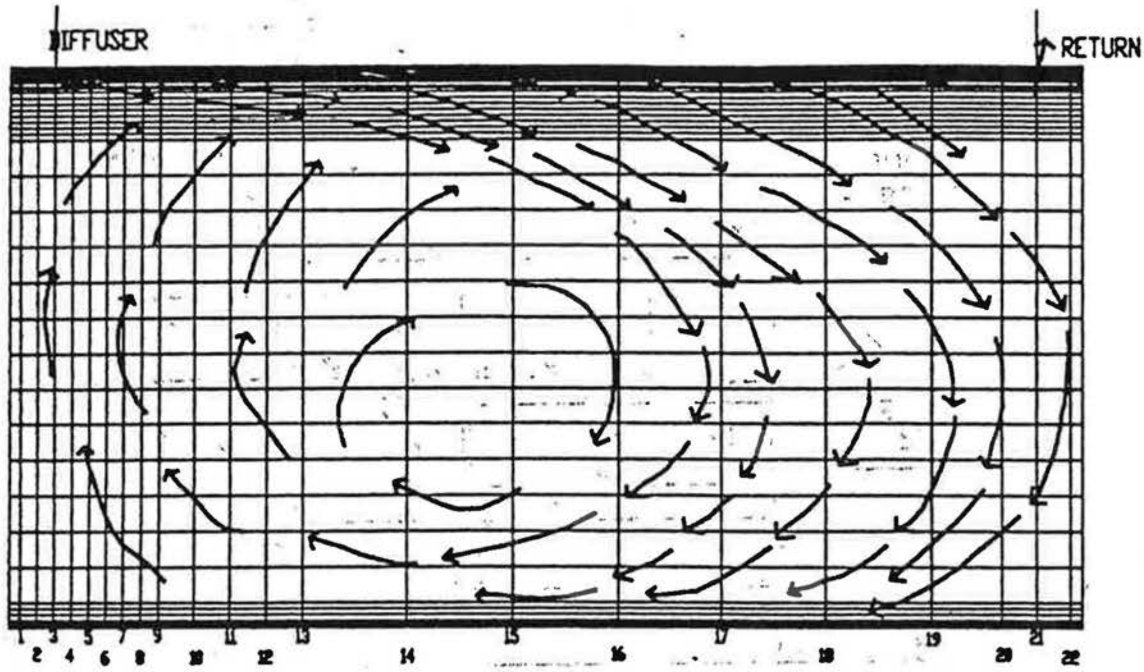

Figure 2 Measurement grid and visualized airflow patterns in test case 1

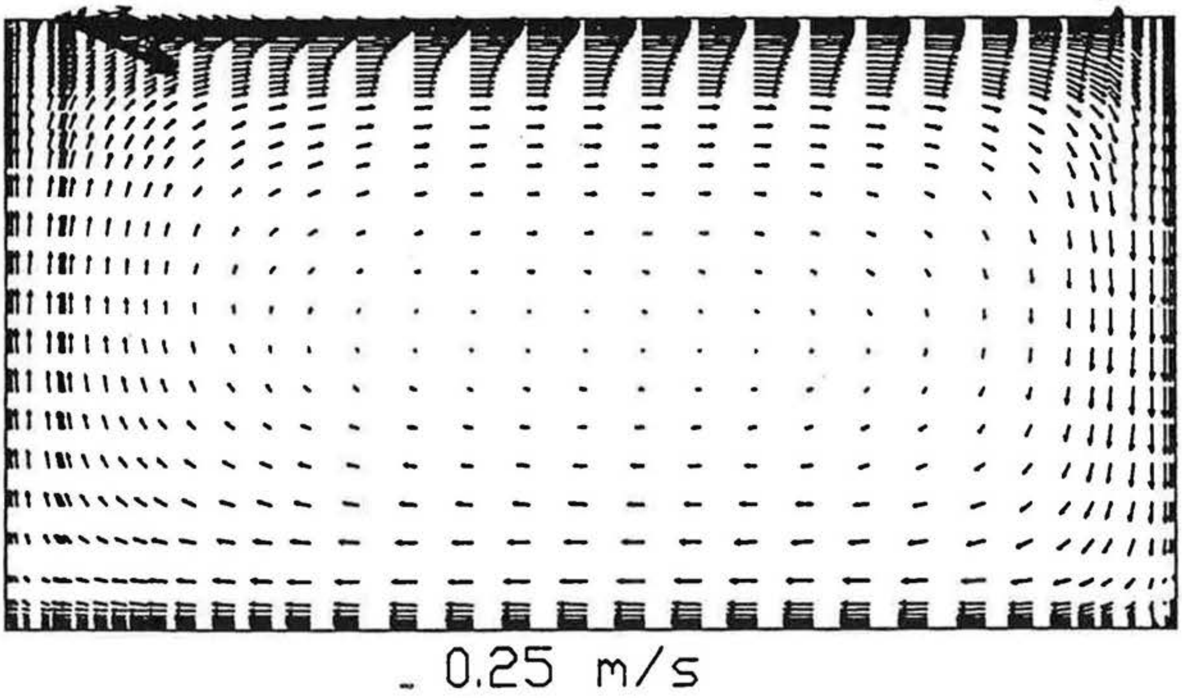

Figure 3 Computed airflow pattern at the symmetric plane in test case 1

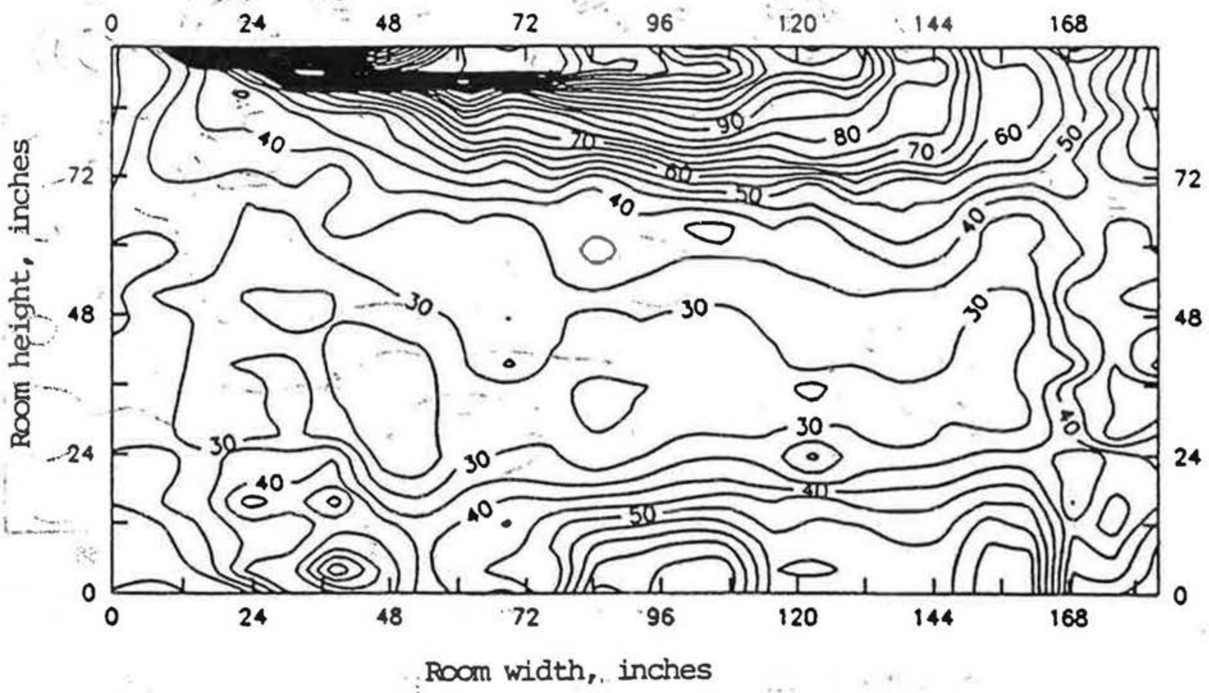

Figure 4 Measured air velocity (ft/min) distribution at the symmetric plane in test case 1

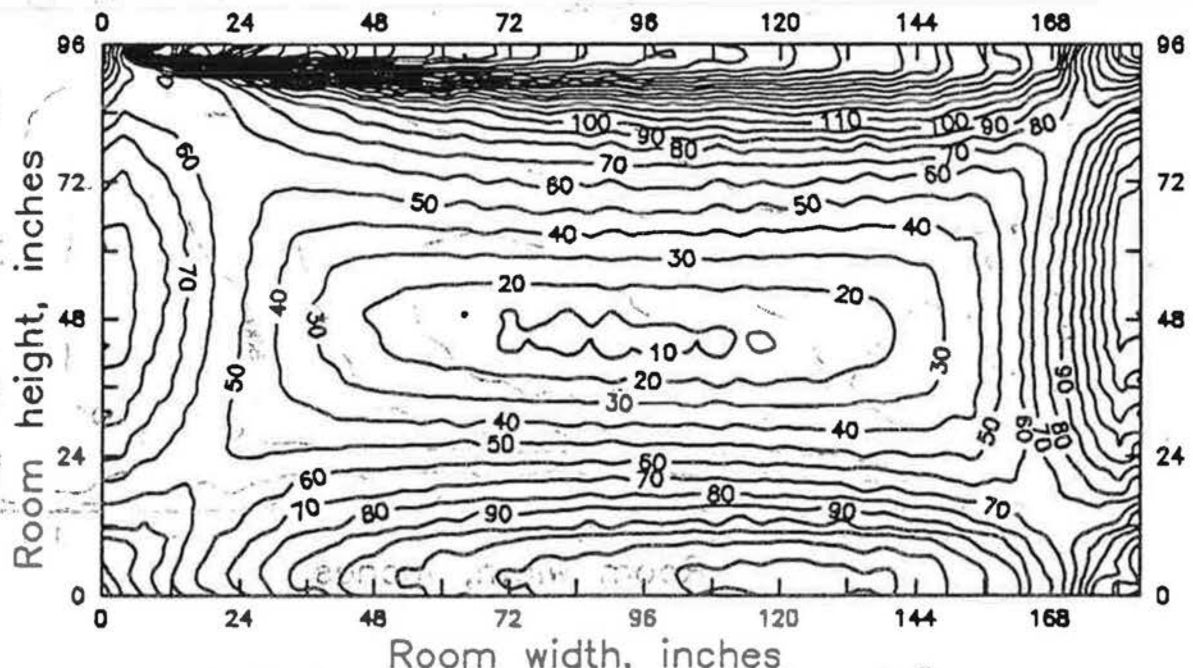

Figure 5 Computed air velocity (ft/min) distribution at the symmetric plane in test case 1

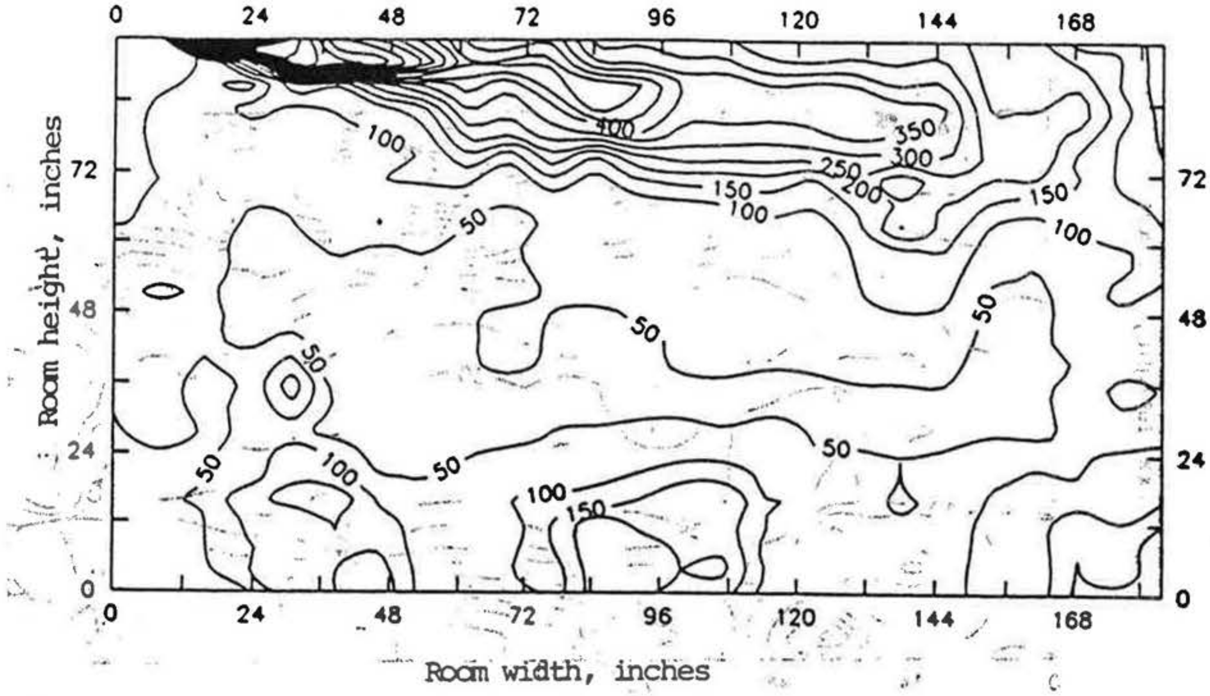

Figure 6 Measured turbulent kinetic energy distribution [(ft/min)^2] at the symmetric plane in test case 1

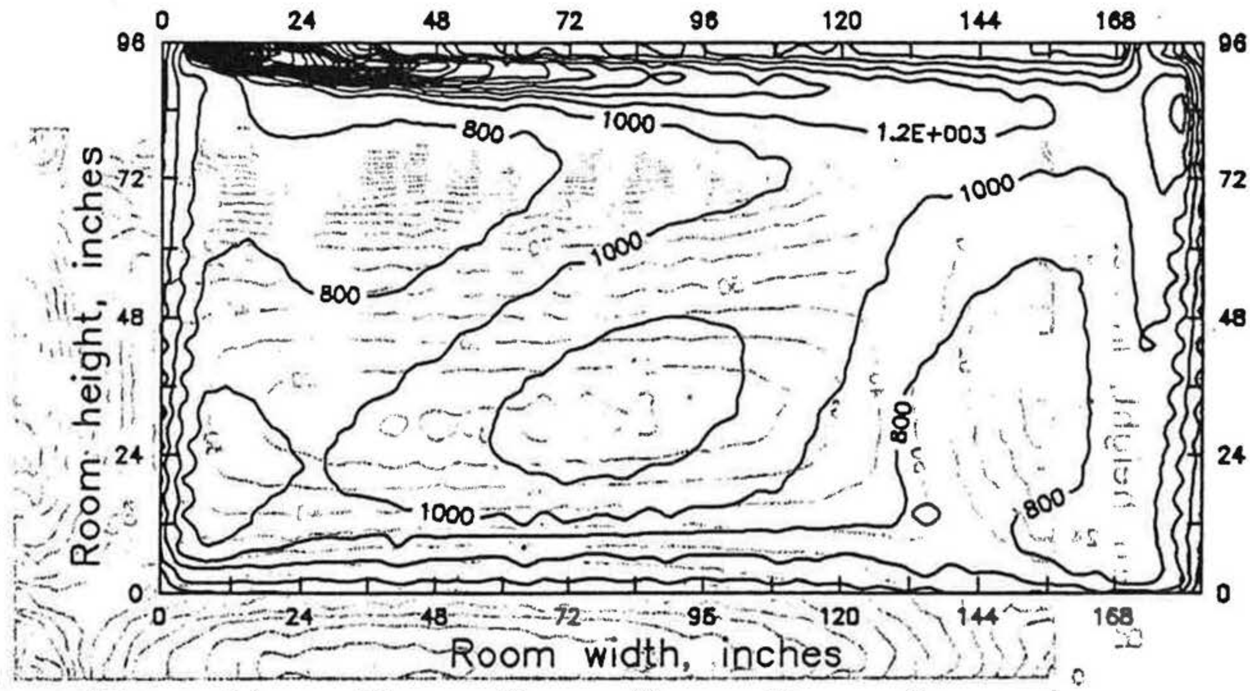

Figure 7 Computed turbulent kinetic energy distribution [(ft/min)^2] at the symmetric plane in test case 1

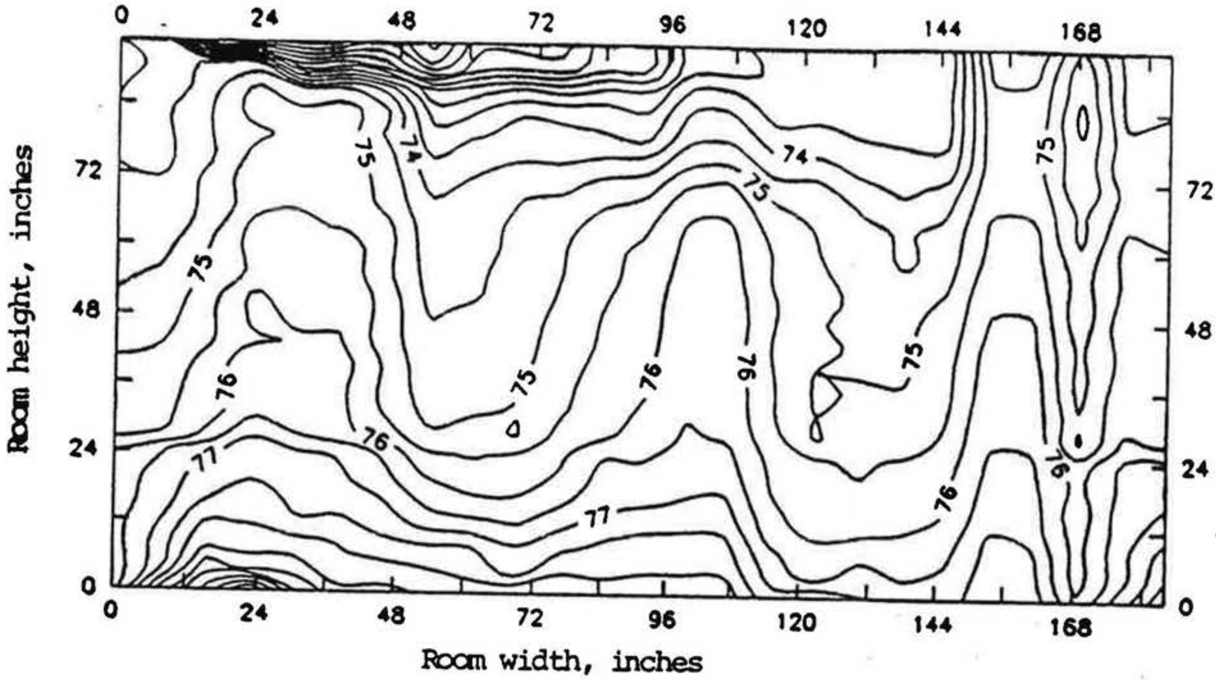

Figure 8 Measured temperature distribution (F) at the symmetric plane in test case 1

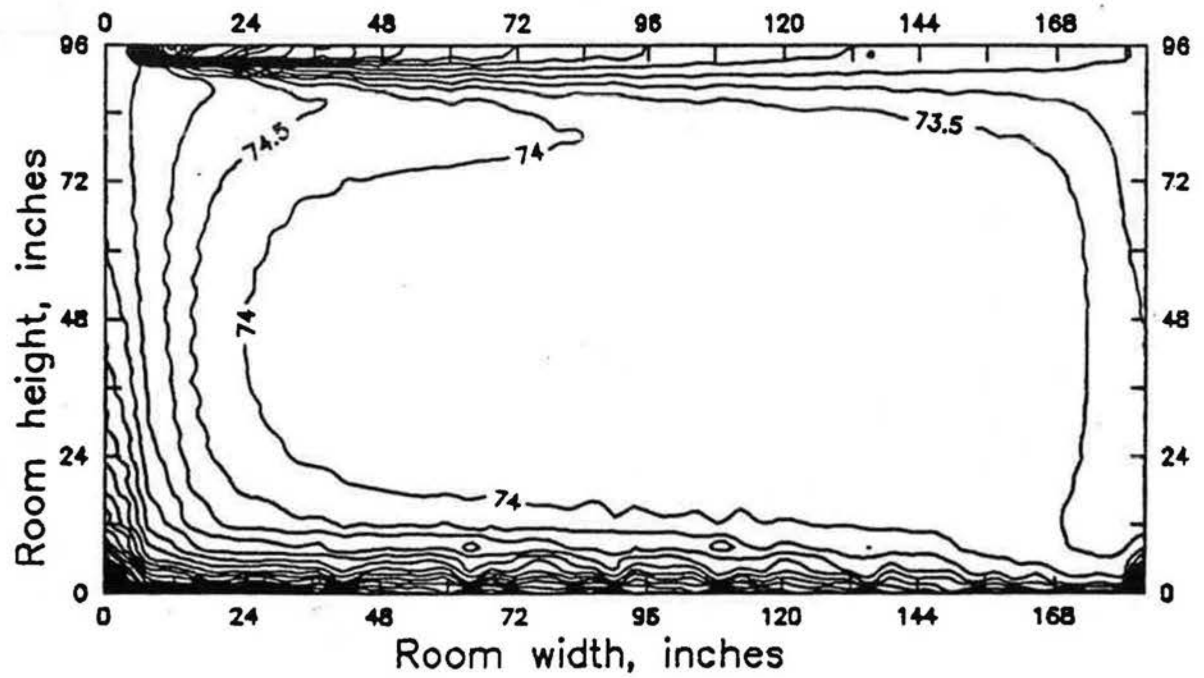

Figure 9 Computed temperature distribution (F) at the symmetric plane in test case 1