4085

Natural Convection in Passive Solar Buildings: Experiments, Analysis, and Results*

Ashok Gadgil, Fred Bauman, and Ronald Kammerud
Passive Solar Analysis and Design Group
Lawrence Berkeley Laboratory
Berkeley, CA 94720

Abstract *Computer programs have been developed to simulate numerically natural convection in two- and three-dimensional room geometries. The programs have been validated using published data from the literature, results from a full-scale experiment performed at the Massachusetts Institute of Technology, and results from a small-scale experiment performed at LBL. One of the computer programs has been used to study the influence of natural convection on the thermal performance of a single zone in a direct-gain passive solar building. It is found that the convection heat transfer coefficients between the air and the enclosure surfaces can be substantially different from the values assumed in the standard building energy analysis methods, and can exhibit significant variations across a given surface. This study implies that the building heating loads calculated by standard building energy analysis methods may have substantial errors as a result of their use of common assumptions regarding the convection processes which occur in an enclosure.*

Nomenclature

A	Aspect ratio = H/L
C_p	Specific heat at constant pressure
Gr_L	Grashof number, = $g.\beta.\Delta T.L^3/\nu^2$
g	Acceleration due to gravity
H	Enclosure height
L	Enclosure length
$\overline{Nu}_L$	Average Nusselt number, $=gL/\Delta T\kappa$
$\overline{Nu}_{L_{IN}}$	Average Nusselt number on the hot wall
p	Dimensionless pressure
Pr	Prandtl number, = ν/α
q	Heat flux
Ra_L	Rayleigh number, = $Gr_L \cdot Pr$
$\widehat{Re}$	Dimensionless scale of time
T	Temperature
ΔT	Characteristic temperature difference, $= T_H - T_C$
ΔT^*	Maximum temperature on hot wall – minimum temperature on cold wall
t	Dimensionless time
T_{AIR}	Average temperature of air in the zone
T_{MRT}	Mean radiant temperature in the zone
T_C	Average temperature of the cold wall
T_H	Average temperature of the hot wall
$\overline{V}$	Dimensionless fluid velocity factor
X	Vertical axis (dimensionless)
Y	Horizontal axis (dimensionless)
α	Thermal diffusivity = $\kappa/\rho C_p$
β	Coefficient of volumetric expansion
δ_i^j	Kronecker Delta, = 0 if $i \neq j$; 1 if $i = j$
θ	Dimensionless temperature
κ	Thermal conductivity
μ	Viscosity
ρ	Density
ν	Kinematic viscosity = μ/ρ

Introduction

In spite of the fundamental role it plays in both conventional buildings and passive solar systems, natural convection has received relatively little experimental or analytic attention within the building sciences. Within a single thermal zone†, natural convection and thermal radiation are jointly responsible for the distribution of heat from collection and/or storage media to the building occupants, to the occupied space, or to building elements having significant thermal mass but which do not receive direct sunlight. In general, several thermal zones are needed to characterize occupied buildings (e.g., buildings other than warehouses, airplane hangars, etc.); and convection processes contribute to the thermal transfers among these zones. Buoyancy-driven† convection is responsible for the air circulation producing a cooling effect in sunspaces, multistory atria, and in other thermal chimney designs. Finally, many passive solar concepts such as double envelope structures, thermocirculation systems, and air collection systems rely almost exclusively on natural convection for their operation.

Recent research results have emphasized the importance of natural convection processes. Analyses performed by the Los Alamos Scientific Laboratory (LASL) on the Balcomb house [1] have implied the importance of convective coupling of thermal zones as compared to radiative and conductive cou-

* This work was supported by the Research and Development Branch, Passive and Hybrid Division, Office of Solar Applications for Buildings, U.S. Department of Energy, under Contract No. 7405-ENG-48.

† See Glossary of Technical Terms.

plings in a multizone structure. Preliminary results from work performed at Lawrence Berkeley Laboratory (LBL) [2] indicate that the magnitude of the convection heat transfer coefficients on the inside surfaces of a typical direct gain building configuration can vary significantly with time. This result is consistent with an earlier study [3] which demonstrated that appreciable errors in prediction of building thermal loads can result from the common assumption that the total (convective + radiative) heat transfer coefficients are constant with time.

There is evidence that convective heat transfer processes are highly dependent on both the geometric configuration of the structure being studied (e.g., Ref. 2) and the range of thermal boundary conditions that might be encountered in the structure. Also, the natural convection processes that occur in passive systems are largely uninvestigated. Thus, there is a need to provide a sound technical basis for estimating the effects of convection on the performance of buildings. An unmanageably large number of experiments would be required to explore thoroughly natural convection phenomena in buildings empirically. The present study addresses this problem by focusing on the development of a general computerized numerical method for analysis of natural convection, and on the validation of the method using results from a few selected experiments. The computer code can then, with some confidence, be applied to a broad range of studies of natural convection in buildings. More specifically, the work reported here consists of (1) the development and validation of a numerical analysis technique for studying convective heat transfer in buildings and (2) the use of this analysis technique to study the quantitative role of natural convection in the thermal performance of a direct solar gain structure, and thereby to examine the accuracy of standard assumptions regarding convective heat transfer within a zone in a building.

Background

Past natural convection research [4] has dealt primarily with geometric configurations that do not typify rooms in buildings; as a result, these studies are of limited application in the building sciences. Convection heat transfer coefficients most often used in building energy analysis are largely based on experiments conducted 25 years ago [5-7]. This work was necessarily limited in the range of experimental parameters investigated. In addition, the lack of large computers and sophisticated experimental hardware prevented the researchers from thoroughly examining the sensitivity of their results to the experimental assumptions. This past research has not been extended, most likely due to three factors: (1) The historically low cost of energy used in buildings; (2) the emphasis on the use of forced convection wherever possible; and (3) the difficulty of conducting analytical, numerical, or experimental investigations of configurations representative of buildings.

More recently, there has been renewed interest in natural convection in the building sciences. Investigations of convective heat transfer within and between thermal zones have been reported by Buchberg [3], Nielsen [8], Honma [9], and Weber [10].

Problem Definition

In the published literature, the problem of natural convective heat transfer in an enclosure is typically simplified to the configuration illustrated in Fig. 1. In a two-dimensional rectangular enclosure, one vertical surface is maintained at a constant temperature T_H, and the opposite vertical surface is maintained at a lower constant temperature T_C. The horizontal surfaces are adiabatic (perfectly insulated). This was one configuration chosen for the numerical and experimental comparisons with published data. Heat input or extraction through one vertical surface of an enclosure is a reasonable model for many situations arising in buildings; for example, it may represent heat gain from an unvented Trombe wall or heat loss through windows in single- and multistory buildings. In addition, a previous study [11] indicates that in warm climates the heat losses through the walls and windows (the vertical surfaces) are larger than the losses through the floor and the ceiling (horizontal surfaces) in a well-insulated, single-story, residential building.

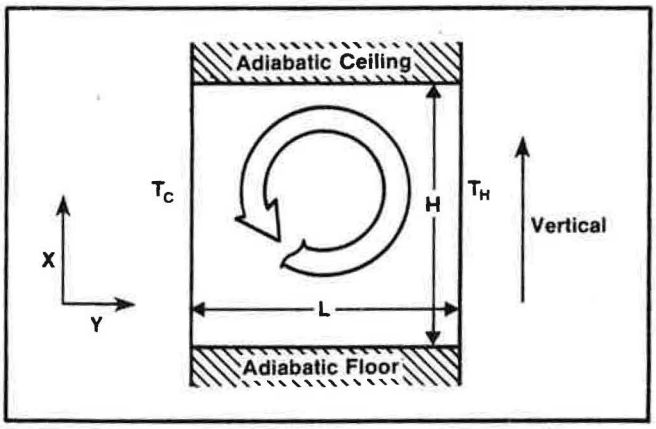


Fig. 1. Recirculating flow induced in a fluid inside a two-dimensional square cavity, defined by adiabatic floor and ceiling and isothermal walls, at tempertures T_H and T_C ($T_H > T_C$).

In the configuration illustrated in Fig. 1, variations in density drive the enclosed fluid up the heated wall, along the top horizontal surface, down the cooled wall, and along the bottom horizontal surface, completing the convective loop. The convective motion of the fluid is confined mostly to a thin region along all four internal surfaces, producing a rather large and fairly inactive central core region. Characteristics of the flow such as the mean air temperature, convection coefficients between air and walls, flow velocities, etc., are completely determined by specification of the three independent dimensionless parameters listed here:

- Aspect ratio: $A = H/L$ where H = enclosure height and L = enclosure length.
- Prandtl number: $Pr = \nu/\alpha$ where ν = kinematic viscosity and α = thermal diffusivity.
- Rayleigh number: $Ra_L = Gr_L Pr = g\beta\Delta T L^3 Pr/\nu^2$, where Gr_L = Grashof number, g = acceleration due to gravity, β = coefficient of thermal expansion, and ΔT = characteristic temperature difference = $T_H - T_C$.

These parameters include all relevant information regarding the enclosure geometry, the fluid properties, and the relative strength of buoyancy and viscous forces, respectively. For a rectangular room twice as long [5.5 m (18 ft)] as it is high [2.75 m (9 ft)] filled with air at 21° C (70° F) and with at least a 1° C (1.8° F) temperature difference between vertical walls, these parameters take the values:

$$A = 0.5,$$
$$Pr = 0.71, \text{ and}$$
$$Ra_L \geq 1 \times 10^{10}.$$

Analysis Description and Comparison with Published Data at Low Rayleigh Numbers

Little numerical work has been published on natural/buoyant convection at Rayleigh numbers in excess of 10^7. In this flow regime, fluid velocities become relatively large. If the popular Central Difference Scheme (CDS) is used for casting the equations governing the fluid flow into finite difference form, the large velocities necessitate an impracticably fine mesh size to ensure numerical stability of the solution procedure (e.g., Ref. 12). Spalding [13] has proposed a differencing scheme that overcomes this difficulty; it allows relatively coarse grid spacings without seriously compromising accuracy and solution stability [14]. Two computer programs, CONVEC2 and CONVEC3, were developed based on this differencing scheme. These programs respectively solve the coupled two- and three-dimensional conservation equations with the Boussinesq approximation:

Continuity: $\text{div}(\vec{V}) = 0,$

Momentum:

$$\hat{Re}\ \frac{d\vec{V}}{dt} + (\vec{V}\cdot\vec{\nabla})\vec{V} = \nabla^2\vec{V} - \text{grad}\ p + Gr\,\theta_i^3,$$

Energy:

$$\hat{Re}\ \frac{\partial\theta}{\partial t} + (\vec{V}\cdot\text{grad})\,\theta = \frac{1}{Pr}\nabla^2\theta.$$

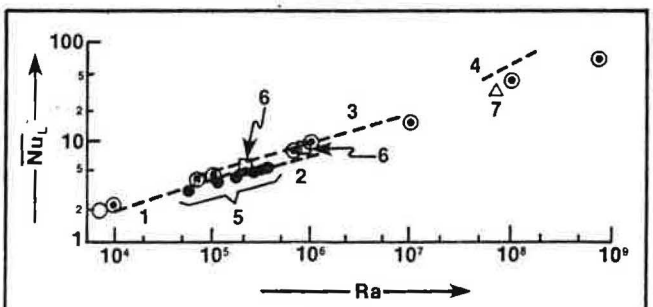
These computer programs can be applied to fluid flow problems driven by predefined temperature distributions on the enclosure surfaces and/or by pressure differentials between specified locations on the boundary. To date, a turbulence model has not been incorporated into either computer program, so the analyses are limited to steady (laminar) flows.* (For a more detailed description of the analysis technique, see Ref. 15 and references cited therein.)

Validation of the two computer programs CONVEC2 and CONVEC3 has been undertaken by comparing the calculated results to various published numerical and experimental efforts and to two recent experiments utilizing room geometries. The comparison to the low-Rayleigh-number data cited in the literature is described below; validations at the higher Rayleigh numbers characteristic of buildings are described in the next sections. The mesh sizes used for calculation in all validations were relatively coarse (the finest two-dimensional mesh size was 17 × 20). The grid lines were distributed evenly throughout the central region (interior of the fluid volume) with a concentration of grid lines near the enclosure boundaries; this permitted simulation of the sharp changes in flow properties associated with a developed boundary layer. Sensitivity analyses indicated that it was adequate for this purpose to position three grid lines parallel and adjacent to each enclosure surface.

A quantity of particular importance in the problem defined by Fig. 1 is the magnitude of convective heat transfer, measured by the Nusselt number. For a square enclosure ($H = L$ in Fig. 1), the average Nusselt number can be defined as:

$$\overline{Nu}_L = \frac{1}{T_H - T_C}\int_0^1 \left.\frac{\partial T}{\partial Y}\right|_{Y=0} dX$$

In Fig. 2, $\overline{Nu}_L$ calculated with the convection code, CONVEC2, is plotted as a function of Rayleigh number. On the same graph, relevant numerical and experimental results for $10^4 \leq Ra_L \leq 10^9$ from various investigators [16-22] have been superimposed; as shown, the agreement is quantitatively acceptable. Additional validation at low Rayleigh numbers has been presented in Ref. 15.



Curve **1:** de Vahl Davis, Calculations [16]
Curve **2:** Emery, Experimental Results [17]
Curve **3:** Portier et al., Calculations, Pr = 0.7 [18]
Curve **4:** Burnay et al., Data Reduced from Experiments, Pr = 0.7 [19]
Result **5:** Rubel and Landis, Calculations [20]
Result **6:** Quon, Calculations [21]
Result **7:** Fromm [22]
Points indicated by ⊙ are from present calculations.

Fig. 2. Dependence of $\overline{Nu}_L$ on Ra, for two-dimensional flow inside a square cavity. Comparison with published results shows acceptable agreement.

Experiments and Analysis Validation at High Rayleigh Numbers

Existing experimental data have largely been limited to Rayleigh numbers of less than 10^9 — at least an order of magnitude below that which characterizes

* For a room-shaped enclosure, buoyancy-driven convection will not become fully turbulent for Rayleigh numbers less than about 10^{11}. The wind-driven convection in a building is always turbulent.

full-scale building geometries. In addition, most of the data are for large aspect ratios, typifying fluid flow in narrow vertical channels. Two recent experiments [2, 23] have expanded the data base into the geometric and kinematic region of interest to the buildings sciences. The experiments are described here and their results are compared to the predictions of the convection analysis code.

Small-Scale Experiment

A small-scale experiment, coordinated with the analysis, is being carried out at Lawrence Berkeley Laboratory. The results from the first phase of this experiment are reported here.

A schematic cross-sectional view of the experimental configuration is shown in Fig. 3; the apparatus has inside dimensions of 12.7 cm (5 in.) high by 25.4 cm (10 in.) long and extended to a horizontal depth of 76.2 cm (30 in.) to minimize three-dimensional effects. Water was used as the working fluid; this allowed representative Rayleigh number values to be obtained in a small-scale apparatus. The range of parameters covered by this experiment are:

$$A = 0.5;$$
$$2.6 \leq Pr \leq 6.8; \text{ and}$$
$$1.6 \times 10^9 \leq Ra_L \leq 5.4 \times 10^{10}.$$

The opacity of water to thermal radiation implies that the experiment was not a direct, scaled approximation to a real room; using water in the experiment allowed measurement of the purely convective part of the heat transfer process being studied* and from this standpoint, was ideal for validating the convection analysis code.

In experimental modeling of convective heat transfer processes, the detailed behavior of a fluid with Prandtl number less than 1.0 cannot be accurately simulated with a working fluid having a Prandtl number much greater than 1.0 [24]. Therefore, the magnitude of the Nusselt number measured in this experiment is not numerically identical to that for an air-filled enclosure. However, the general fluid behavior and parametric relationship observed in the experiment can be expected to be representative of an air-filled cavity.

The heat transfer data obtained from the experiment are presented in the form of $\log_{10}(\overline{Nu}_{L_{IN}})$ vs. $\log_{10}(Ra_L)$ in Fig. 4. In this figure $\overline{Nu}_{L_{IN}}$ (Nusselt number) is a measure of the strength of the convective heat transfer at the heated wall. On the same figure, experimental results from a study by MacGregor and Emery [24] and predictions from an analytic study by Raithby et al. [25] are shown; the present experiment is in quantitatively good agreement with both of these previous results. (Note that at the high Rayleigh numbers shown in the figure, the Nusselt number is relatively insensitive to the aspect ratio [26, 27].)

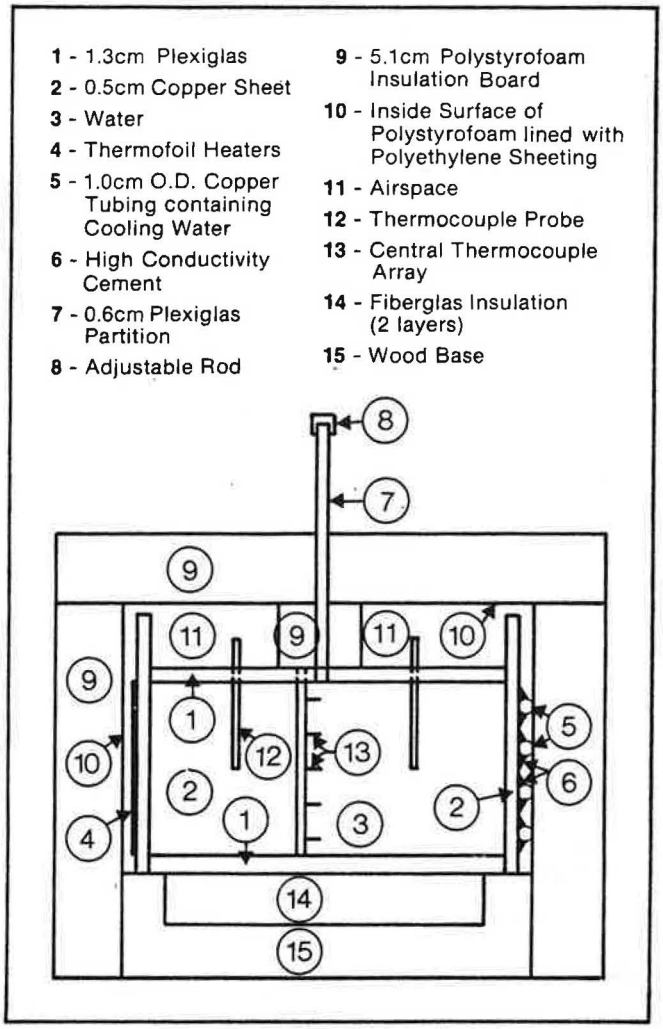


Fig. 3. Schematic cross-sectional view of experimental configurations.

To use the data from this experiment for validation of the two-dimensional convection code, the computer program was modified to incorporate the temperature dependence of the physical properties of water. Sensitivity studies using the computer program demonstrated an increase of as much as 10 percent in the Nusselt numbers when the properties were allowed to vary with temperature rather than being fixed at their average values. Due to lack of sufficiently detailed experimental instrumentation, best estimates of some enclosure surface temperature profiles were necessary to complete the input to the analysis program. These surface temperature estimates are believed to have errors of less than ± 10 percent. To date, the sensitivity of the prediction of the computer code to these uncertainties has not been thoroughly investigated, pending the availability of data from an improved small-scale experiment in progress at LBL.

Comparisons of the predictions of the computer code with the experimental results for the extent of stratification in the core region are shown in Figs. 5 and 6. These figures show the temperature profiles along vertical and horizontal lines through the geometric centers of the enclosure for $Ra_L = 2.4 \times 10^9$ and $Ra_L = 4.7 \times 10^{10}$, respectively. The excellent agreement of the numerical prediction of the temperature profiles in the central core at the lower Rayleigh

* The maximum contribution of thermal radiation to the measured Nusselt number was calculated to be less than 2 percent.

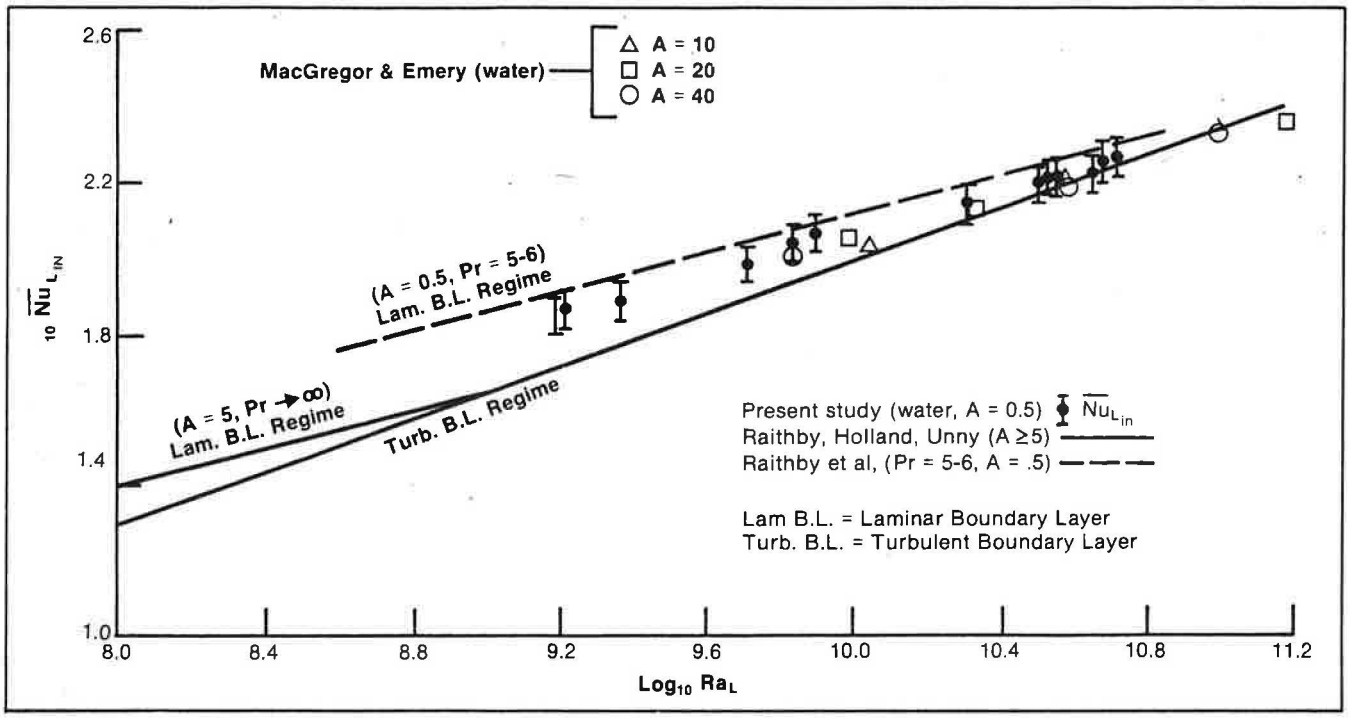


Fig. 4. Heat transfer results compared to previous results.

number (Fig. 5) indicates that the program is addressing the fundamental characteristics of the flow successfully at this Rayleigh number (2.4×10^9). The numerically predicted vertical centerline temperature profile at the higher Rayleigh number (4.7×10^{10}, Fig. 6) exhibits a shift to smaller temperature gradients in the central core region associated with an increased gradient near the horizontal surfaces. The most likely sources of this discrepancy are (1) the potential for transitional flow (between steady laminar and fully turbulent) at this value of Ra_L could contribute to the noted differences between experimental and numerical results; (2) due to increased convective effects at this higher Ra_L, the errors in the estimates of surface temperatures immediately upstream from the centerline region may have a significant effect on the magnitude of the calculated temperature profile at the centerline; or (3) the coarse mesh size used in the present numerical studies may also have been a contributing factor to the disagreement at this high value of Ra_L. Ongoing work is expected to shed some light on this question; until the source of the discrepancy is understood, however, the comparison of stratification profiles implies that the convection code properly represents the fundamental character of the flow for the smaller value of Ra_L and is qualitatively correct for all $Ra_L < 5 \times 10^{10}$.

The Nusselt number predictions made by the computer code are compared to the corresponding measurements in Table 1; the surface temperature distributions used in the simulations are also indicated in this table. The two numerical simulations (runs 2a and 2b) made at the higher Rayleigh number indicate the sensitivity of the predicted Nusselt number to differences in the details of a surface temperature specification. The disagreement in the Nusselt numbers at the lower Ra_L value is thought to be due to the sensitivity of the Nusselt number predic-

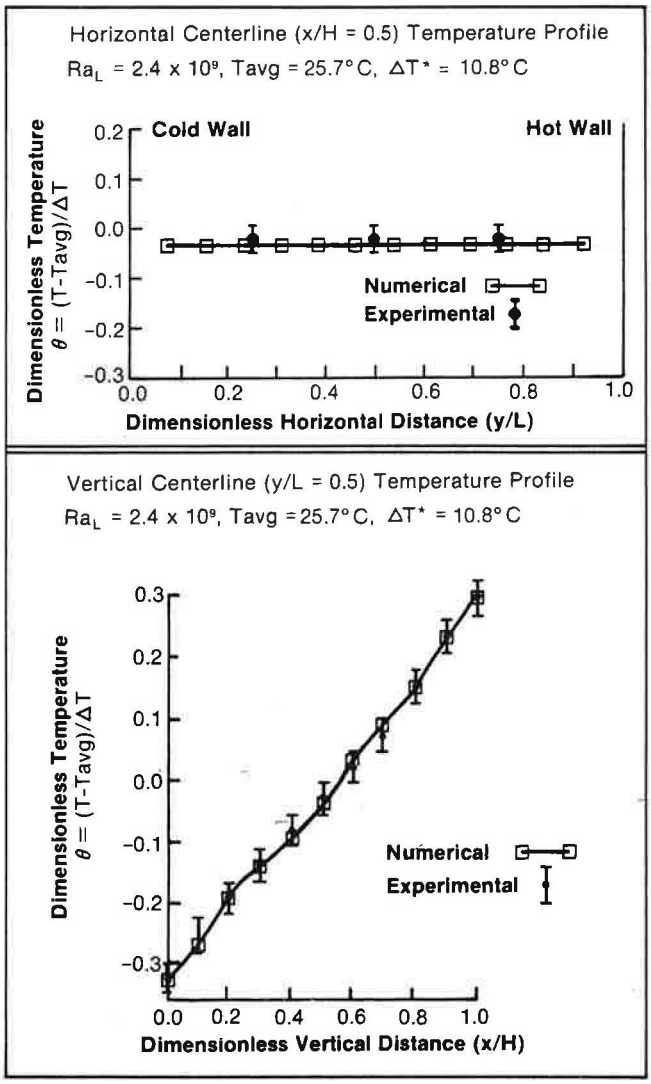


Fig. 5. Comparison of code predictions to experimental results for the extent of stratification in the core region for $Ra_L = 2.4 \times 10^9$.

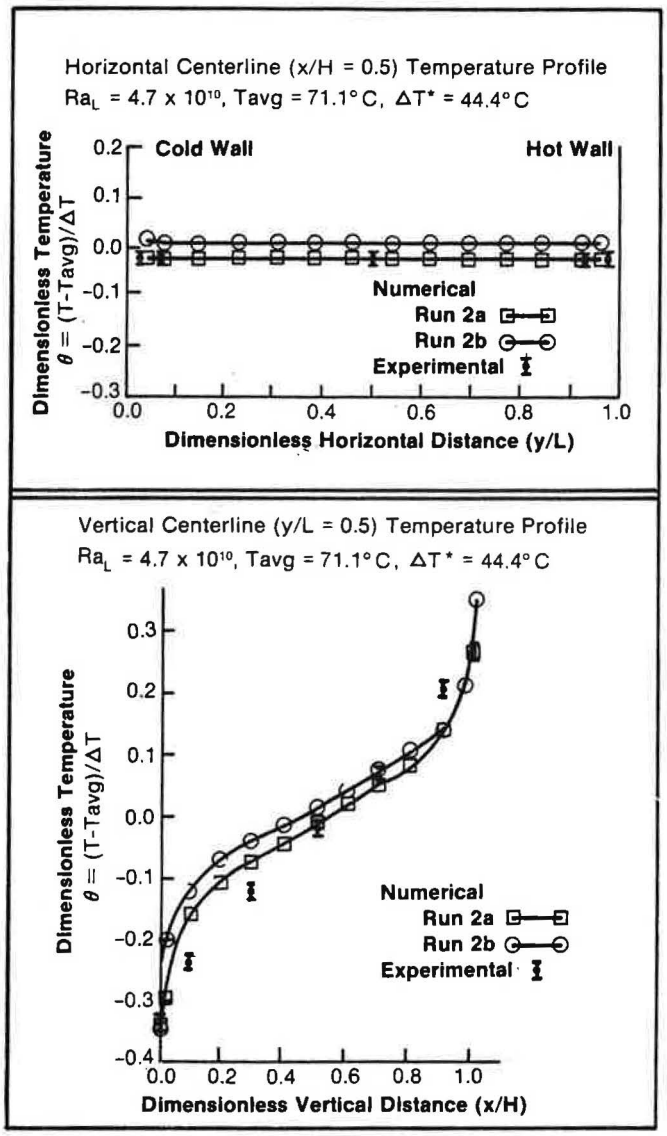


Fig. 6. Comparison of code predictions to experimental results for the extent of stratification in the core region for $Ra_L = 4.7 \times 10^6$.

Run	Ra_L	Experimental	Numerical	Surface Temperatures for Numerical Simulations
1	2.4 x 10⁹	79 ± 12	105	Estimated from Experiment
2a	4.7 x 10¹⁰	165 ± 12	168	Estimated from Experiment
2b			201	Hot & Top Wall = T_H Cold & Bottom Wall = T_C

Table 1. Comparison of Hot Wall $\overline{Nu}_{L_{HN}}$.

tion to the details of the (unmeasured) surface temperature distributions.

A Full-Scale Experiment

Natural convection was investigated by Ruberg [23] at the Massachusetts Institute of Technology in the full-sized test room shown schematically in Fig. 7. The conditions of this experiment correspond more closely to those in a real building. Heat was supplied

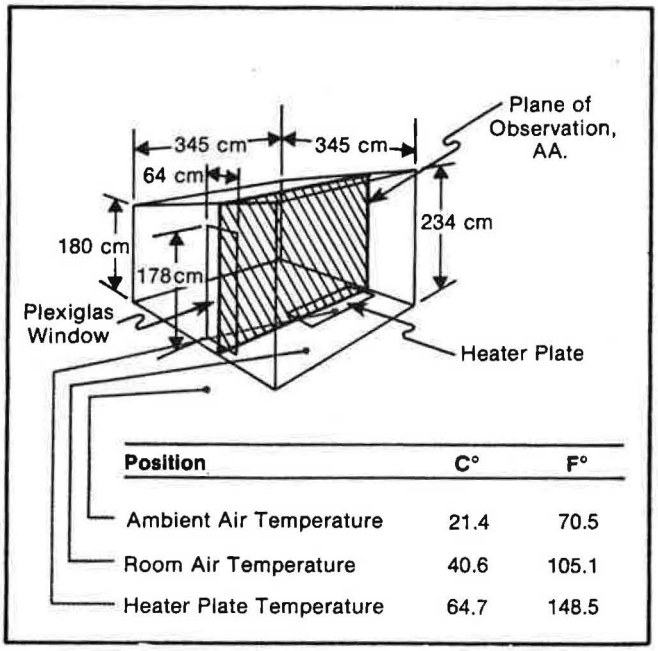


Fig. 7. Schematic diagram of Ruberg's full-scale test [23].

to the test room by an electric resistance heater plate on the floor. A single-pane window was located on one wall of the enclosure. Though the window represents only 2 percent of the envelope area of the test room, it dominated the thermal load; 22 percent of the total heat loss was measured to be through this surface. Steady-state conditions were maintained by controlling the temperature of the air outside the test room to within ± 0.2° C (0.4° F) of its average value. The construction was sufficiently airtight that the effects of infiltration could be ignored.

The heater plate configuration was intended to represent a solar irradiated area on the floor; its rectangular shape and its size [1.16 m² (12.5 ft²)] were well matched to those of the 1.12-m² (12-ft²) window, and its heating capacity [448 W/m² (142 Btu/hr ft²)] was selected to approximate solar radiation falling on a dark-colored floor with low thermal capacity at noon on a clear day at 40° N latitude. The parameters for this experiment were:

$$A = 0.58,$$
$$Pr = 0.71, \text{ and}$$
$$Ra_L = 5 \times 10^{10}.$$

Convection observed in the test room was characteristic of the transition regime between laminar and turbulent flow. A schematic of the air flow patterns is shown in Fig. 8. Note the essentially three-dimensional character of the flow.

Air temperatures were measured with a vertical array of 11 thermistors mounted on a motorized boom which traversed the test room in the measurement plane. The array had 20-cm (8-in.) vertical spacings in the center and 10-cm (4-in.) spacings near the ceiling and floor; data were recorded at 20-cm (8-in.) horizontal intervals as the boom moved across the room. This resulted in a grid of 11 × 18 temperature data points in the measurement plane which perpendicularly bisects the window and the heater surfaces (plane AA in Fig. 7).

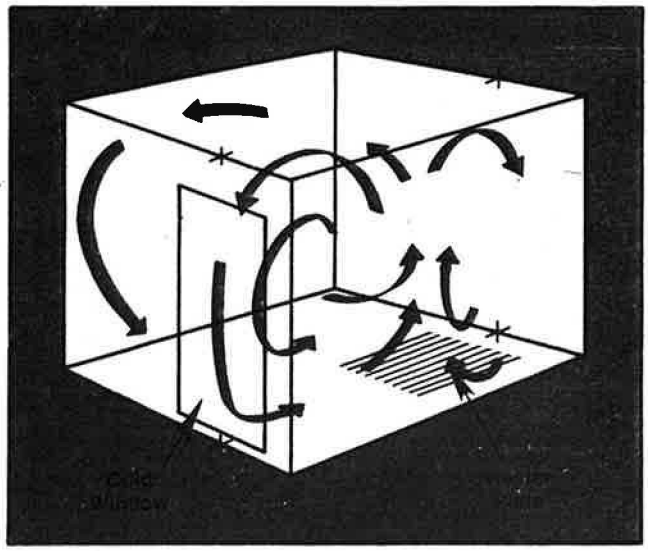


Fig. 8. Schematic of air-flow patterns in Ruberg's experiment [23].

Temperature measurements were converted to isotherms separated by intervals of 5 percent of the maximum surface-temperature difference in the room. The isotherms were referenced to the mean temperature of the room, indicated on Fig. 9 as 0.000. The three sets of isotherms represent data from three separate measurements of air temperatures under identical conditions. From the isotherms, the air flow patterns shown in Fig. 8 may be discerned: the plume over the heater plate, a layer of warm air at the ceiling, a cool air current at the window and along the floor, and a somewhat isothermal area in the center of the room. For more details of this experiment, see Ref. 23.

Data from this full-scale experiment were compared with the predictions from the three-dimensional version of the numerical code. The surface temperature profiles used in the analysis were estimated from the available data using a thermal balance technique. It is noted that the effects of radiation on the temperature probes are expected to bias the measured air temperatures toward higher values by an unknown amount. This effect was not accounted for in the thermal balance. This bias also affects the comparisons between the isotherms predicted by the numerical scheme (Fig. 10) and the experimental results (Fig. 9). Also, as noted above, the observed flow was in the transition regime (between laminar and fully turbulent), but was simulated with a numerical code assuming laminar flow. In light of these limitations, the agreement is satisfactory.

The validations described here have been performed with the available experimental data covering the range of interest of the important dimensionless parameters. The discrepancies between the predictions and the observations are thought to be understood, at least qualitatively. The computer program appears to simulate the convective flow correctly for $Ra_L < 5 \times 10^{10}$. The second phase of the ongoing small-scale experiment at LBL is expected to provide quantitatively a sound basis for further validation of the computer code at higher Rayleigh numbers.

The computer program, in its present validated form, has been used to study the influence of natural

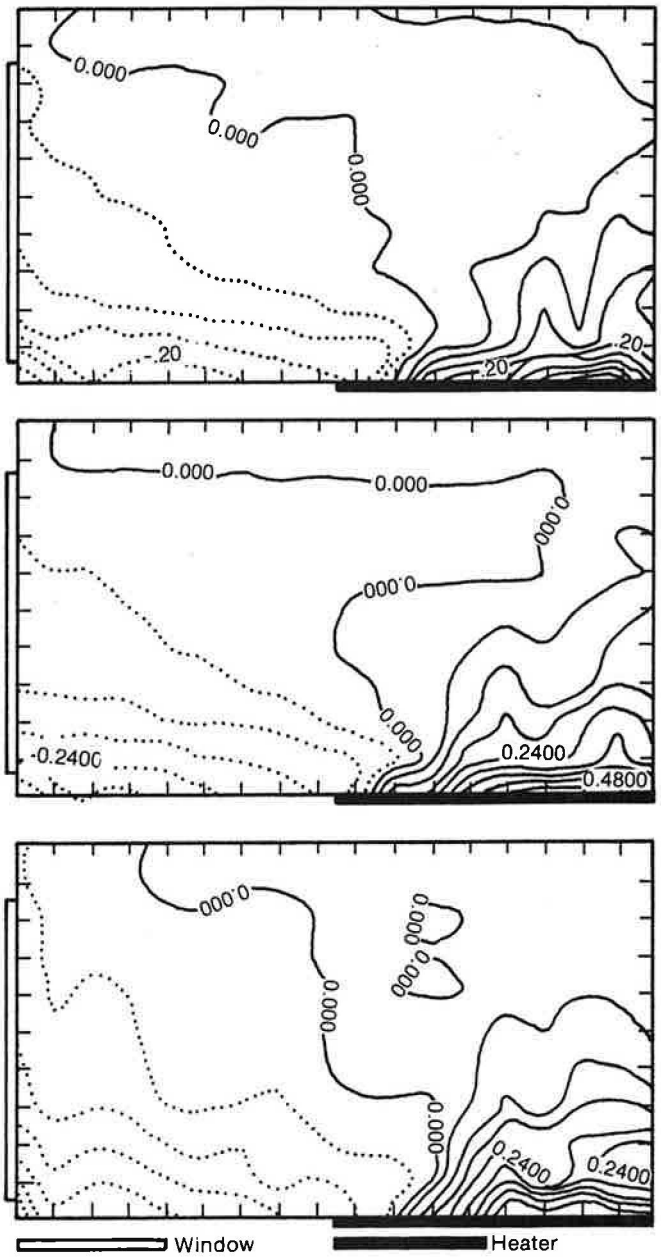


Fig. 9. Isotherms measured in Ruberg's experiment [23].

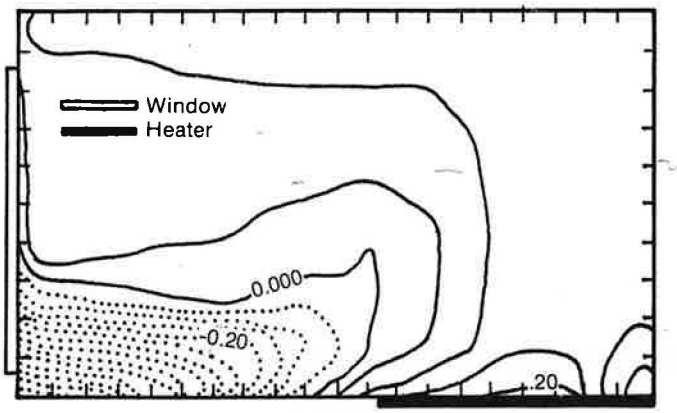


Fig. 10. Isotherms predicted by three-dimensional numerical code.

convection on the thermal performance of a single room in a direct gain passive solar building. This is described in the next section.

Convection Effects in Building Energy Analysis

Most building energy analysis techniques, including the most sophisticated computer codes (BLAST*, DOE-2†, etc.) and other passive solar system analysis programs, make the simplifying assumptions that (1) the air temperatures throughout the volume of the individual zones in the structure are uniform, and (2) the convection heat transfer coefficients for the surfaces of the building being analyzed are constants. These assumptions are largely consistent with the state of knowledge regarding convection at the time the codes‡ were developed. In a preliminary study [15], it was shown that convection coefficients at the surfaces of an enclosure are actually quite sensitive to the temperature distributions on the surfaces. To estimate the effects of this observation on the accuracy of results from the programs, the convection code was used iteratively with BLAST to obtain self-consistent surface temperature distributions and convection coefficients.**

The south-facing zone (S-zone in Fig. 11b) of a multizone building was studied. The floor plan of the building is shown in Fig. 11a. Figures 11b and 11c show the thermal zones used in the BLAST simulations. The building has been thoroughly described elsewhere [28]. This zone has dimensions of 3.7 m wide × 9.2 m long × 2.5 m high (12 ft × 30 ft × 8 ft). For the purpose of this study, the following modifications to the zone were made. The interior of the analyzed zone was made up of 14 surface segments. The two "end surfaces" (the east partition wall and the west exterior wall) measured 2.5 m × 3.7 m (8 ft × 12 ft) and were very highly insulated. The other four major surfaces (ceiling, slab floor, gypsumboard north partition wall, and the south exterior wall) were each divided into three equal subsurfaces (see Fig. 12). The individual subsurfaces extended the full 9.2-m (30-ft) length of the zone. The middle section of the south wall was specified as double-pane window. Iterative analyses were performed for both the nighttime (loss) and daytime (gain) modes for a selected winter design day.

For each iterative sequence of calculations, BLAST was first used to calculate the surface temperatures of each subsurface defining the zone for each hour of the design day using the standard constant convection coefficients. BLAST performs a full thermal balance on all surfaces of the zone under study and the zone air. The surface thermal balance accounts for thermal radiation between zone surfaces; convection between zone air and each surface; conduction through each surface; and radiative gains from occu-

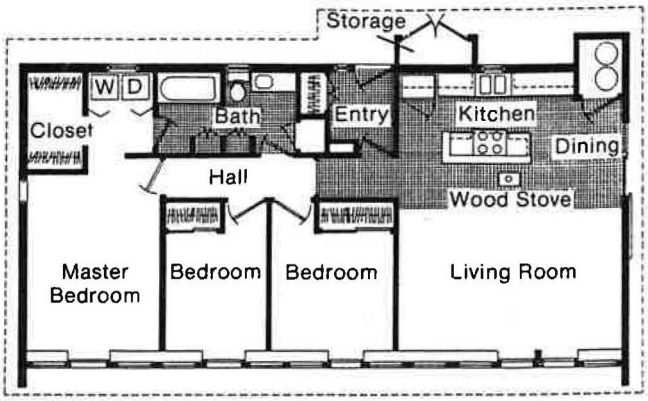


Fig. 11a. Floor plan of test building.

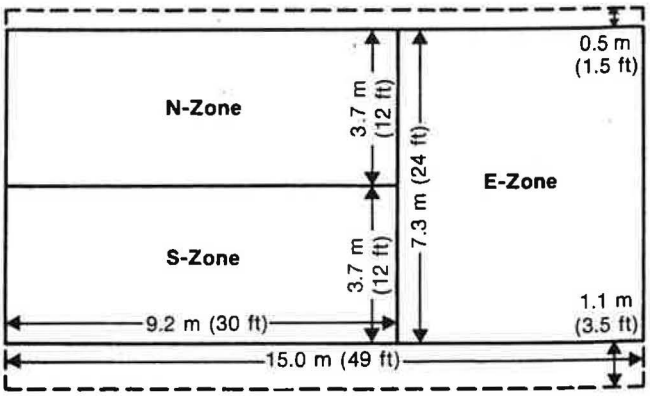


Fig. 11b. Floor plan showing thermal zones for BLAST simulation.

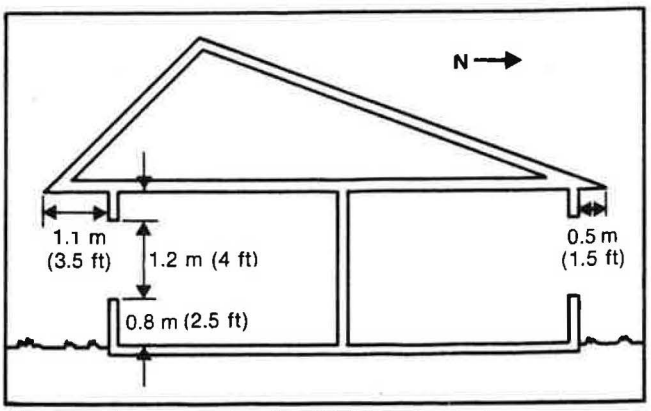


Fig. 11c. Cross-section for BLAST simulation.

---

* BLAST (Building Loads Analysis and System Thermodynamics) is copyrighted by the Construction Engineering Research Laboratory, U.S. Department of the Army, Champaign, Illinois.

† DOE-2 is a public domain program being developed by the Division of Communities and Building Energy Systems, U.S. Department of Energy.

‡ Some of the codes do utilize convection coefficients for nonvertical surfaces which are sensitive to the direction of heat flow, but not to the magnitude of the temperature difference between the room air and surface or to the possibly large effects induced by the differences in the temperatures of the different surfaces defining the zone.

** The convection heat transfer coefficient (CHTC) for a given surface is defined by the relation CHTC = $q/\Delta T$ where q = heat flux from the surface into the room air, ΔT = (average surface temperature) - (room air temperature).

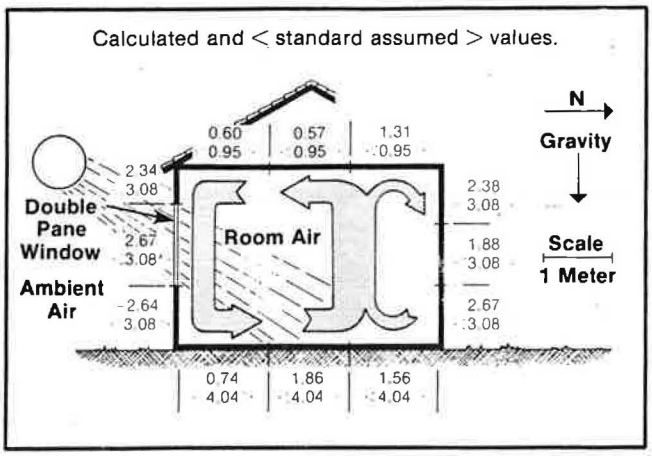


Fig. 12. Convection coefficients (W/m² °C) between the room air and the interior subsurfaces of a single zone in a house. Steady state (daytime) heat gain mode.

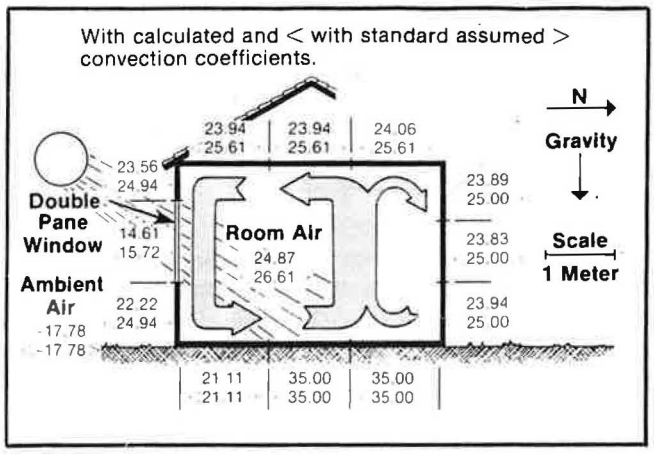


Fig. 13. Surface temperatures (°C) on the interior subsurfaces of a single zone in a house. Steady state (daytime) heat gain mode.

pants, lights, equipment, and transmitted solar energy. The thermal balance on the air accounts for convective gains from surfaces, occupants, lights, and equipment and for controlled and uncontrolled ventilation. For this study, the relevant output from BLAST was the distribution of temperatures of the subsurfaces defining the zone boundary. From the design day results, two hours were chosen for further analysis of convection: one hour at night when the zone is in the loss mode and one hour during midday when the zone is in the solar gain mode.

Because of the zone geometry and the distribution of the surface temperatures, the convective flow was expected to be two-dimensional. The subsurface temperatures calculated by BLAST were input to the two-dimensional convection code to simulate the details of the convection process and to calculate convection heat transfer coefficients for each subsurface. These coefficients were then used as input to BLAST to obtain the new subsurface temperatures. These temperatures were again used as input to the convection code, and the entire procedure was iterated until self-consistent results were obtained. Several features of the iteration process should be noted here.

- At this time, the convection code cannot account for sources and/or sinks of thermal energy in the air volume under study. Therefore, the BLAST simulation did not include auxiliary heating and/or cooling of the zone in which the convection was to be analyzed; the zone temperature was free-floating though adjacent zones were heated. This limitation led to the selection of an exterior dry-bulb temperature for the design day such that the zone air temperature would float near a typical daytime thermostat (nighttime thermostat setback) temperature. These temperatures are shown in Figs. 13 and 15.
- During the loss mode (nighttime) iterations, the north partition wall, representing a warm storage wall, and properly accounting for the existence of a conditioned zone to the north, was held at a constant temperature and was the primary heat source for the zone. During the gain mode (daytime), only the two subsurfaces of the slab floor closest to the north partition wall were irradiated by solar transmission through the south glazing. This configuration corresponds to midday conditions during the winter (solar altitude = 30°). These subsurfaces were the primary heat source for the zone, and their surface temperatures were held constant throughout the iteration process. Here, too, the effect of the adjacent conditional zones was properly accounted for by the BLAST analysis. The glass was also held at a constant surface temperature during the gain mode.
- A design day with varying ambient temperatures and other environmental parameters was used initially to calculate the starting points for the iteration scheme, but during the iterations steady-state external conditions were assumed to observe convergence more clearly. Since the BLAST simulations assumed no auxiliary heating or cooling in the zone being analyzed, the exterior temperature was selected to ensure that reasonable comfort conditions were maintained in the zone with the specified constant surface temperatures described.

The results of the detailed convection analysis are summarized in Figs. 12-15. The surface temperatures and convection coefficients obtained both with and without the iterative procedure using the convection code are shown in these figures. More detailed heat transfer data for the two modes are given in Tables 2 and 3. In these tables, subsurfaces are numbered sequentially around the zone; results for the thermal parameters for each subsurface appear in the table. Case I refers to standard assumptions, and Case II refers to the values after the iterations.

The convection coefficients are seen to change substantially from their standard assumed values for most of the surfaces. This is particularly true during the loss mode in which strong boundary-layer flows develop along both the warm north wall and the cold south window. During the gain mode, circulation induced by large, warm areas of the floor does not contain such strong boundary-layer flows. The balance point air temperature (mean radiant temperature) for the zone is observed to change by 0.51°C

Surface	Subsurface Number	Subsurface Location	Standard Building Energy Analysis Assumptions: Surface Temp. °C	Convection Coefficient W/m² °C	Convective Heat Flux From Zone To Surface W/m²	Total Heat Flux From Zone To Surface W/m²	Values After Iteration With Convection Code: Surface Temp. °C	Convection Coefficient W/m² °C	Convective Heat Flux From Zone To Surface W/m²	Total Heat Flux From Zone To Surface W/m²
South Exterior Wall	1	Top	20.11	3.08	2.53	7.46	19.72	4.27	2.99	6.91
	2	Middle/Window	11.22	3.08	29.91	83.87	10.06	1.97	20.41	77.18
	3	Bottom	20.11	3.08	2.53	7.46	18.61	-1.79	-3.24	7.03
Slab-On Grade Floor	4	South	20.89	4.04	0.16	0.57	20.39	7.18	0.22	0.28
	5	Middle	20.89	4.04	0.16	0.57	20.39	2.73	0.08	0.14
	6	North	20.89	4.04	0.16	0.57	20.39	1.64	0.05	0.11
North Partition Wall	7	Bottom	26.67	3.08	-17.68	-51.91	26.67	2.47	-15.44	-52.91
	8	Middle	26.67	3.08	-17.68	-51.91	26.67	1.60	-10.02	-47.49
	9	Top	26.67	3.08	-17.68	-51.91	26.67	1.15	-7.20	-44.67
Ceiling To Attic	10	North	20.56	0.95	0.35	2.67	20.17	8.90	2.22	3.55
	11	Middle	20.56	0.95	0.35	2.67	19.83	1.17	0.69	3.98
	12	South	20.56	0.95	0.35	2.67	19.83	0.85	0.50	3.79
			T_{AIR} = 20.93, T_{MRT} = 20.96				T_{AIR} = 20.42, T_{MRT} = 20.40			

T_{AIR} = Average Temperature of Air in the Zone, °C T_{MRT} = Mean Radiant Temperature in the Zone, °C

Table 3. Convective Analysis of a Single Zone, Loss Mode (Figs. 14 and 15)

would be nearly twice as rapid as that from the top portions. Another interesting feature is the very large convection coefficient [8.9 W/m²°C (1.56 Btu/hr ft²°F)] for the portion of the cool ceiling directly warmed by the updraft from the warm north wall. Since the ceiling was well insulated, the temperature difference between this portion of the ceiling and the room air was decreased (by about 50 percent) with only a negligible change in the heat loss to the attic.

The results presented in Table 3 show that during the gain mode, the convective heat transfer from the floor to the air is reduced by about a factor of two when correct convection coefficients are used. This contributes to the observed lowering of the air temperature in the zone. Total heat transfer to the nonilluminated massive north partition wall is reduced by only approximately 20 percent. Also, losses from the south window are reduced by about 8 percent; this change is about equally split between reductions in convection and radiation.

During both modes of operation, the portion of the south wall directly below the double-pane window encounters a downdraft of cold air that has lost heat through the window. This downdraft of air is actually colder than the interior surface of the bottom portion of the south wall (this surface is warmed by radiative exchange with the other surfaces of the zone). Thus, although the bottom section of the south wall is actually cooler than the average room temperature, it deposits heat into the cold downdraft flowing across it. This has resulted in a negative convection coefficient for this subsurface (the convection coefficients are defined with respect to the average room air temperature).

From a practical viewpoint, convection coefficients and surface temperatures are of little real interest except as they influence comfort levels and/or thermal load. Here, the building load is a quantity of fundamental interest. To estimate the effect on the thermal load of a change in any building parameter, the accepted "exact" method (i.e., coheating) is described as follows: The building temperature is maintained at a constant level before and after the parametric change by introducing a heat source/sink of appropriate magnitude. The difference between the heat supplied/removed by the source/sink in the two cases gives the effect of the parametric change on the building load [29].

Since the BLAST simulations described here did not include heat sources/sinks, a less exact method based on the balance point air temperature was used to estimate the effect of the changed convection coefficients on the zone load. This method was calculated (using a radiation balance technique) to be in error by less than 2 percent (with respect to the results of the "exact" coheating method) for the configuration under study.

The zone under examination was simulated by BLAST with two different thermostat control profiles. The "base case" thermostat settings were (arbitrarily) set at 21.1°C (70°F) for daytime (gain mode) heating and 16.7°C (62°F) for nighttime (loss mode) heating. The base simulation assumed the same external weather conditions as used in the iteration procedures for the two modes of operation. Iterations with the convection code had predicted changes in the balance point air temperatures of 1.7°C (3°F) and 0.5°C (1°F) for the gain and loss modes respectively. The net effect of the balance point change would be to decrease the cooling load and/or increase the heating load. Therefore, during the second load calculation the second thermostat profile was set at