PREDICTION OF SMOKE MOVEMENT IN BUILDINGS

Edward E. Zukoski
California Institute of Technology
Pasadena, California 91125

Introduction

The purpose of this paper is to describe briefly and in a general way several schemes used to calculate the motion of smoke through a structure or part of a structure involved in a fire. Calculations of this type are required if we are to understand the spread fire through the structure and if we are to learn how to write building codes and other regulations which can provide the best possible protection for building occupants. For example, one of aims of designers of high rise buildings is the incorporation in their designs of regions where people can be protected from contact with combustion products without the requirement that they be able to escape from the building. Another aim is to restrict combustion products to the floor of the fire origin at least and, if possible, to a part of that floor. Finally, consider the simpler problem posed by the requirement that a ceiling or wall of a burning home be opened to change the flow of smoke such that firemen can approach the fire. To obtain a solution of each of these problems, we must be able to calculate the effects of smoke control measures (active or passive) on the gas motion.

In the following sections we shall first discuss in a general manner several schemes for calculating the flow of gas produced by externally applied pressures. This external field can be produced by the heating and ventilation system, the stack effect, or winds and need not depend on fire-induced natural convection effects.

We then turn to a discussion of several experimental studies of flames driven by natural convection effects and to a model of the flow produced by a small fire in a single room. The aim of this last section is to illustrate the nature of some of the current research work concerning natural convection problems and the detailed modeling of a single-room fire.

No attempt is made here to summarize the work going on elsewhere in this broad field. Documents which summarize the present state of the art and present details of some of the many computer codes being developed are listed in References 1 - 3..

Motive Forces

A number of processes are available to set up a flow of gas through a complex building. The most obvious is the heating and ventilation system which is designed to recirculate air through the building with volume flow rates often large enough to produce rapid changes of the air in the whole structure. Pressure differences of a few tenths of an inch of water are typically used. This system also could be used for active control of combustion product flows during a fire. One of the aims of the models discussed below is to determine how such control capabilities can best be used.

Two other processes, the stack effect and wind-induced pressures, can cause air to move air within the building even if density differences within the structure are negligible. The stack effect is the name given to the hydrostatic pressure difference produced when the temperature inside and outside the building differ appreciably. For example, the pressure difference between a point at the first floor level and at an upper floor level L feet higher of a building is $\rho_b gL$ inside the building, whereas it is $\rho_o gL$ outside the building. (Here, ρ_b is the gas density inside and ρ_o is the density outside, and g is the gravitational constant.) Thus, if pressures are equalized at a first floor door, a pressure difference $(\rho_o - \rho_a)gL$ would be available to produce an outflow through any opening in the exterior wall of the upper floor. Should an outflow occur in the upper floor levels, a corresponding inflow will be induced in the lower levels to maintain a pressure balance between the floors of the building. Thus, a circulation can be set up within the building. For large temperature differences between the interior and exterior of the building, say $35^{\circ}C$, in a 100-foot high building, pressure differences of 0.2 inches of water are generated.

Strong winds impinging on the sides of a building will also set up pressure fields which can drive flows through the building. A wind of 20 miles per hour can produce a pressure difference across a structure of up to 0.4 inches of water.

Finally, natural convective forces resulting from fire-heated gases produce pressure differences of the order of 0.05 to 0.15 inches of water in a one-story high structure and larger pressures if more than one story is involved.

Note that the four mechanisms discussed here produce pressure differences of roughly the same size. Of these forces, only natural convection is directly dependent on the heat generated by a fire. However, the others can cause the motion of toxic combustion products even when temperature differences are too small to produce an appreciable motion due to natural convection. Hence, all four must be considered in our discussion.

The flow within the building set up by those mechanisms will depend strongly on the location of the fire, the weather (i.e. temperature and the direction and magnitude of the wind), details concerning the geometries of openings connecting the inside and outside of the structure, and the configuration of the building (i.e., the particular arrangement of doors, ventilation duct dampers, etc. which affect the possible flow paths within the structure). The random nature of these important factors does restrict the utility of the calculations discussed below. However, if the motive forces and the response of the building - gas system to them is understood, a rational approach can be made to many fire protection and fire fighting problems.

Uniform Density Model

The first elaborate computer modeling of flames in a building have dealt with situations in which natural convection was not important (see, for example, ref. 1). In these situations, gas moves through the structure under the influence of a pressure field produced by the stack effect, wind loads, or a heating-ventilating system. Density is constant within the building and enters only through a hydrostatic pressure field. The principal problem here is to describe the flow rate through every opening in the building and to match these flows with the imposed pressure conditions.

Consider a single opening across which the pressure difference is ΔP. The resulting volumetric flow rate through the opening, $\dot{Q}$, is given by an expression of the form

$$\dot{Q} = \left(2 \frac{\Delta P}{\rho}\right)^n CA \quad ,$$

where n is a number with value of 1/2 for large openings and 1 for flows in which viscous losses are dominant; C is a flow coefficient which characterizes the geometry of the opening; A is the area of the opening; and ρ is the density of the gas, a constant here. Each opening in the building to be considered in the calculations must have values of n, C, A specified.

For a steady flow problem, the sum of all the volumetric flows through all the openings of a "room" must be zero to satisfy the conservation of mass. The pressure distribution throughout the whole structure can be found by requiring that conservation of mass be satisfied in each room of the structure in such a manner that the imposed pressure conditions are satisfied.

In a complex structure, some simplification can be gained by treating a number of rooms, halls, and vertical shafts as a single constant-pressure region. If shafts or stair wells which extend many floors are considered, pressure differences within such a region are easily calculated by assuming that hydrostatic equilibrium exists and that local density is determined by the temperature characterizing the whole interior of the structure. That is, a local stack effect calculation can be made.

When the ventilation system or wind loads are important it is necessary to specify the dependence of the applied pressure field on the volumetric flow rates through various duct openings. That is, the pumping characteristics for the driving mechanism must be specified.

Clearly, in this model of the flow field, the primary difficulty is to manage a very large number of openings and to correctly specify the opening coefficients and pumping characteristics. In addition, it is also possible with little increase in complexity to extend the analysis to treat the movement of toxic tracer gases through the structure. Consider again a single room with a number of openings. If we assume that the room can be treated as a region of uniform conditions, then the concentration of a trace species ε (e. g. parts per million or mass fraction of carbon monoxide) will be constant in each region. The mass flow rate of this species leaving the room at an opening with an outflow will be $\rho \varepsilon \dot{Q}$ and, similarly, a mass flow rate will enter from adjoining regions through openings with an inflow of gas. Conservation of mass can again be applied to each room or region to find the steady state field for the trace species. Thus, a species conservation equation is added to the overall mass balance equation. The basic assumption made here is that mixing of gases entering the room with the gas within the room is strong enough to produce a uniform distribution of the trace species within the whole region. In the analysis, the effect of the trace element on density is ignored, but could be included in calculating the hydrostatic pressure field. The source strength of the trace species must be specified as one of the boundary conditions.

No conceptual mathematical difficulties arise in extending this trace gas analysis to a non-steady problem in which, for example, the source strength is allowed to vary. In this problem, the conservation of mass equations must be modified to include a varying value for species concentration in the individual regions. If the species in question is present in small amounts, this transient calculation can be superimposed on a steady flow.

The principal problem with a model of this type is that it is not useful when natural convection forces are important, as they will be near a fire. It is most useful in describing the motion of toxic gases present in small amounts in regions "far" from the fire and for assessing the response of the ventilation system to perturbations produced by the fire.

Constant Room Properties

A major modification of the analysis must be made if heat addition or species source terms are strong enough to make natural convection effects of appreciable size compared with other driving forces. Under these conditions the density differences between gas in adjacent regions or rooms may be large enough to affect the flow rates across the connecting opening.

As an example, consider first adjacent rooms with different temperatures and densities for which the pressure difference due to the imposed pressure field is comparable to the hydrostatic pressure difference produced by the density difference acting over the height of the opening. Analysis, to be discussed later, shows that, depending on the imposed pressure difference, flow in either direction can occur across the opening and that for a narrow range of pressure differences, flow in both directions can occur simultaneously.

The mass flow rate or rates across the opening will now depend on both the pressure and density differences. However, if the assumption that each region is perfectly mixed is made, then the type of approach suggested for the toxic gas tracers can be used again here. The principal modifications are the use of a more complex form of flow equation at each opening, and an equation for conservation of energy.

Thus, for each room, we can generate equations for conservation of mass and energy which depend on a mass flow rate equation or equations for each opening. Given the pressure and heat and/or species input conditions, the flow field and temperature/species fields in the building can be determined.

As before, the analysis can be extended to cover transient behavior resulting from time-varying heat input rates, the losses of energy to room walls by simple heat transfer approximations, etc. Finally, the behavior of the fire itself must be included to model fire spread in any complete transient model. However, under many examples of interest, a quasi-static computation of the temperature, species, and flow fields for a specified state of the fire will be sufficient to answer many important questions concerning protection of escape routes, the temperature and toxic gas levels in various rooms, and the effects of active control efforts on smoke movement in a building.

More Detailed Modeling

The most limiting assumption made above is that each region of the building is treated as a perfectly mixed region in which temperature and species concentration must be taken to be constant. This assumption is reasonable for a room in which the fire is well developed (after "flashover") and adjacent regions, but certainly has limited applicability during early stages of the development of a fire or for rooms far away from regions of strong heat input. For these cases, strong stratification will exist within any one region or room, and this stratification must be taken into account to adequately describe the flow field. The description of the effects of stratification on movement of hot, toxic gases within a building at this level of detail is currently the subject of a number of research projects (e.g. ref. 2). To illustrate the problems being studied, I will present a brief description of work being carried out on such problems at the California Institute of Technology under support from the National Science Foundation and the National Bureau of Standards. More details concerning these studies are given in several progress reports (refs. 4 and 5).

To understand the problems we are studying, it is useful to consider the development of a room fire in the fire - room - hall - shaft arrangement shown schematically in Figure 1. In the upper sketch, the fire is in an early stage of development; a thermal plume has formed above the fire, has impinged on the ceiling where it spreads out to form a layer of hot gas, called the ceiling layer. At this stage, ambient air is forced to flow out of the door due to the density reduction of the gas inside the room resulting from heat input from the fire. Only a small part of the gas in the ceiling layer has taken part in the combustion process; most of it is air which has been entrained by the buoyant plume, and consequently ceiling layer temperatures are only a few hundred degrees centigrade or less.

In the second sketch of Figure 1, the fire has grown slightly, the ceiling layer has reached a greater depth, and has spilled out into the hall. A strong flow of hot gas down the hall ceiling has resulted from this outflow. Considerable mixing of hot and ambient gas occurs at the head of this gravity flow and in the region just outside the doorway where the ceiling layer gas undergoes a process like a hydraulic jump. However, except for these regions and an area in the neighborhood of the buoyant fire plume, little mixing occurs at the horizontal interface between hot ceiling layer gas and ambient gas.

In the third sketch, the room has become completely filled with hot gas as a result of the continued growth of the fire, and the ceiling layer is deep enough that little mixing occurs in the region just outside the door. However, mixing can occur in the doorway between the outflowing and inflowing streams, and this process becomes more important as the ceiling layer interface gets below the middle of the room. The hot gas in the hall ceiling layer has reached the end of the hall and has been reflected; the hot gas in this region has started to mix vigorously with the cold gas above it in the vertical shaft.

Several fluid dynamic aspects of the flow field discussed here cannot be adequately treated with our present understanding of stratified flows. These include the turbulent mixing processes; that is, the fire plume - ceiling layer interaction region, and the mixing within the ceiling layer where the buoyant plume is subjected to a sudden change in buoyancy as it passes into the layer; the mixing in the doorway between "high speed" inflowing and outflowing jets; mixing between the outflowing jet and ambient fluid where the outflowing stream impinges on the ceiling of the hall; and the mixing in the vertical shaft where the higher density gas in the shaft lies above the hot gas in the hall.

The purposes of the research efforts at the California Institute of Technology are to obtain a better understanding of these mixing processes and the convective heat transfer processes that accompany them, and to construct simple models for the whole flow field. Three problems abstracted from the flow field sketched in Figure 1 are discussed briefly in the remainder of this paper. They are: (1) the mixing in the vertical shaft, (2) the propagation of the gravity current down the hall, and (3) a model for the fire room in the stage of fire development shown in the second sketch of Figure 1.

(1) Turbulent Mixing in a Vertical Shaft. Consider a vertical, air-tight tube which is initially filled with fluid of density $(\rho_a + \Delta\rho_i)$ and is surrounded by ambient fluid with a lower density, ρ_a. At the beginning of the experiment, the bottom of the tube is removed. Because the heavier fluid lies above the lighter, the suddenly exposed interface is unstable and turbulent mixing of the two fluids starts. A well-defined front moves up the shaft, and the heavier fluid is gradually replaced by the ambient fluid through a turbulent transport process. Dr. Johnnie Cannon (who studied this process with me) and I believe that this

process is related to that by which hot gas generated by a fire on one floor moves up a stairwell or elevator shaft to higher floors in the absence of substantial stack effect, ventilation-system induced flows, etc. Hence, we felt it was important to understand the scaling laws for such processes.

In a real fire, a sudden burning through of a partition might produce a situation similar to the sudden opening of the shaft used in the experiments described here. However, the gradual supply of hot gas to the bottom of the shaft is a more probable situation, and the present work is being extended to cover this case. In the real fire, heat transfer processes will have an important effect which is absent from the experiments discussed here. This process is also under study.

The experiments suggested to us that this turbulent mixing process could be modeled by a non-linear diffusion equation. The non-linear form of the equation predicts that the initial changes in density will propagate as a definite front, which was observed experimentally. The equation also suggested a form for scaling the time which allowed a reasonable correlation of our experimental results (see ref. 6).

The motion of the initial front is shown in Figure 2 in terms of Z (the length from the tube entrance to the front, normalized by tube diameter d) and a dimensionless time variable

$\sqrt{(\Delta\rho_i/\rho_a)(g/d)}\ t$ (where g is the gravitational constant and t the time measured after the tube opens). The curve and equations shown here are completely empirical, and the data were obtained with brine and water as the working fluids.

To illustrate the times involved if these results are applied to a full-scale fire, consider a ten-foot diameter shaft 100 feet in length ($Z/d = 10$, $d = 10'$) with hot gas temperature 300°C and cold gas 27°C ($\Delta\rho_i/\rho_a \approx 1/2$). We find that a time of about 50 seconds is required for the front to propagate up the shaft.

The subsequent replacement of the higher density fluid in the shaft by the lower density fluid below the opening can be illustrated briefly by describing the change in density at the top of the tube as a function of time. Data of this type are shown in Figure 3. Here, the density difference at the top of the shaft, $(\rho - \rho_a) \equiv \Delta\rho$, normalized by the initial value $(\rho_i - \rho_a) \equiv \Delta\rho_i$, is presented as a function of the dimensionless time modified by the factor $(L/d)^{9/4}$. The correlation of data is excellent and fits the analytic prediction:

$$(\Delta\rho/\Delta\rho_i) = 1/[1 + k(\tau)]^2$$

where k is a single constant which must be determined from the experiments and τ is the dimensionless time.

Scaling was equally successful when argon-air mixtures were used instead of the brine-water mixtures (ref. 6).

Consider again the full-scale situation described above, with $\Delta\rho_i/\rho_a = 1/2$ and $L/d = 10$. The time required to reduce the density difference at the top of the shaft (i. e., to increase the temperature) to half its initial value, $(\Delta\rho/\Delta\rho_i = 1/2)$, was about $3\frac{1}{2}$ minutes. Thus, for this shaft, the front reached the top in about 1 minute and an additional $2\frac{1}{2}$ minutes was required to substantially replace the high density gas with lower density (hotter) gas. These experiments give us a start toward understanding a more realistic problem. Effects of shaft geometry (e. g. stairs) and heat transfer must be included. Preliminary work indicates both are important.

(2) Horizontal Propagation. The movement of hot gas in to a corridor initially filled with cool air shows strong contrasts with the highly turbulent process described in the previous section. Consider a horizontal tube filled with fluid with density greater than that of the surrounding fluid. When one end of the tube is opened, a distinct front is again produced. However, in this example, no mixing occurs between the lower density fluid entering the tube and the higher density flow leaving it; the front moves at a steady velocity given by $k\sqrt{(\Delta\rho_i/\rho_a)gd}$ where k has the value 0.4 to 0.45. The front of hotter fluid fills about the upper half of the duct so that the tube is roughly half filled with

hotter fluid behind the front, see Figure 4. Benjamin (ref. 7) has surveyed the more general problem in which the flow of low density (hot) fluid is restricted. The above velocity is a reasonable estimate unless the hot flow is greatly restricted. For our full-scale example, d = 10 feet and $(\Delta\rho_i/\rho_a) = 1/2$, the velocity is about 6 feet/second; thus, the front would move down a 100-foot long corridor in about 18 seconds. Contrast this with the $3\frac{1}{2}$ minutes or 210 seconds required for the vertical propagation discussed in the previous example. Clearly, horizontal propagation is quite fast as compared with vertical motion.

In both of these problems, the basic modeling of these flow fields is understood, and we are now studying combinations of the basic systems. For instance, in both examples the propagation of the density front can be stopped by producing a flow of the higher density fluid toward the open exit. In the horizontal case, a velocity approximately equal to the propagation rate of the front in the stationary fluid is required.

(3) Room Model. We are interested in developing a model for describing the flow conditions which exist in an early stage of development of a room fire such as that suggested by the middle sketch of Figure 1 and the more detailed sketch of Figure 5. Observations made of one-fourth scale room fires and of modeling experiments with brine - water systems suggest a

number of approximations that allow a simple model to be constructed which still retains the important features of the flow.

We assume that fresh air enters the opening without mixing with the outflowing stream of hot gas. This ambient air is entrained and heated by the fire plume and enters the ceiling layer exclusively through the fire plume. The ceiling layer is treated as a constant temperature region; gas enters from the fire plume and leaves through the top of the opening without further mixing with ambient air.

In order to complete this model we must describe the entrainment characteristics of the plume and flow processes at the opening. Sufficient empirical information is available to prove an adequate description of axisymmetric or line fire plumes when the density differences across the plume are small and elevations above the fire are large compared with fire diameter. We have adapted these results for use in the model.

Flow through the opening has been modeled by a number of authors. Our scheme is as follows: the hydrostatic pressure field set up by the ceiling layer - ambient gas density difference across the vertical walls of the room is calculated in the absence of an opening but with an arbitrary pressure difference at the floor level. A simple orifice flow equation is then used at each point in the opening to calculate a local velocity which this pressure difference would produce at a small hole in the wall located at that elevation and in the absence of other openings. Then this velocity field is integrated over the area of the opening to calculate the mass fluxes. Experimental studies indicate that this process gives a good representation of the mass flow rates needed to complete the model.

The unknown characteristics of this flow are the ceiling layer temperature T_c and interface height Z_c, and the mass flow rates of the stream entering and leaving the room, $\dot{m}_1$ and $\dot{m}_2$. Application of equations of conservation of mass and energy (see Figure 5) allow these quantities to be determined as functions of the opening geometry (width b and height H) and fire heat input rate and geometry (e. g. axisymmetric, line, etc.).

The model outlined here has been extended to cover the non-steady development of the ceiling layer and a room with a single fire but with an arbitrary number of openings with arbitrary pressure differences across each opening at the floor level. The effects of heat conduction to the walls containing the ceiling layer also have been included in the ceiling layer energy balance, and a model of the radiant and convective heat transfer processes is being developed (ref. 4).

The primary element missing from this model is a transient description for the combustion process. The Harvard -

which remain to be solved. One of the more serious concerns the model for flow through an opening. When the ceiling layer interface lies below the middle of the opening, mixing occurs between the inflowing and outflowing jets and becomes more important as the interface is lowered. As the interface decreases well below the bottom of the opening, the fluid filling the lower section of the room becomes contaminated with the ceiling layer gas and can no longer be treated as ambient fluid. Modeling for this mixing process has not been developed but the process is being studied.

Another problem is that of describing the fire plume adequately when the heat addition due to combustion (i.e., the flames) extends up to or above the ceiling layer interface.

Concluding Remarks

A number of models are being used and others are being developed to describe the motion of combustion products through a building. They differ widely in their aim, degree of completeness and detail provided, and number of regions described. An adequate description of turbulent mixing processes involving stratified flows is one of the most serious defects in current models. These models can provide a useful tool in the study of the spread of fire and smoke through buildings and will aid in designing structures which are safer for their occupants.

REFERENCES

1. Movement of Smoke on Escape Routes in Buildings, Proceedings of Symposium No. 4 held at Watford College of Technology, Watford, Herts. on 9th and 10th April, 1969. London: Her Majesty's Stationery Office (1971).

2. Croce, Paul A. "A Study of Room Fire Development: The Second Full-Scale Bedroom Fire Test of the Home Fire Project (July 24, 1974). Volume I. Test Description and Results," Factory Mutual Research Corporation (June 1975), FMRC Serial No. 21011.4 .

3. Harmathy, T.Z. "A New Look at Compartment Fires, Parts I and II," Fire Technology, Vol. 8, nos. 3 and 4 (1972), pp. 196-217 and 326-351.

4. Zukoski, E. E., Kubota, T., and Veldman, C.C. "An Experimental Investigation of the Heat Transfer from a Buoyant Gas Plume to a Horizontal Ceiling. Part I: Unobstructed Ceiling, Part II: Effects of Ceiling Layer," Technical Report No. 1, National Bureau of Standards Grant No. 5-9004 (October 1975), Daniel and Florence Guggenheim Jet Propulsion Center, California Institute of Technology, Pasadena, Calif. 91125.

5. Zukoski, E. E. "Convective Flows Associated with Room Fires," Semiannual Progress Report (June 1975) under National Science Foundation Grant No. GI 31892X1.

6. Cannon, Johnnie B., Jr. "Convective Flows under Conditions Applicable to Fires in High Rise Buildings," Ph.D. Thesis, California Institute of Technology, Pasadena, Calif. (1974).

7. Benjamin, T. Brooke "Gravity Currents and Related Phenomena," Journal of Fluid Mechanics, Vol. 31 (1968), pp. 209-248.

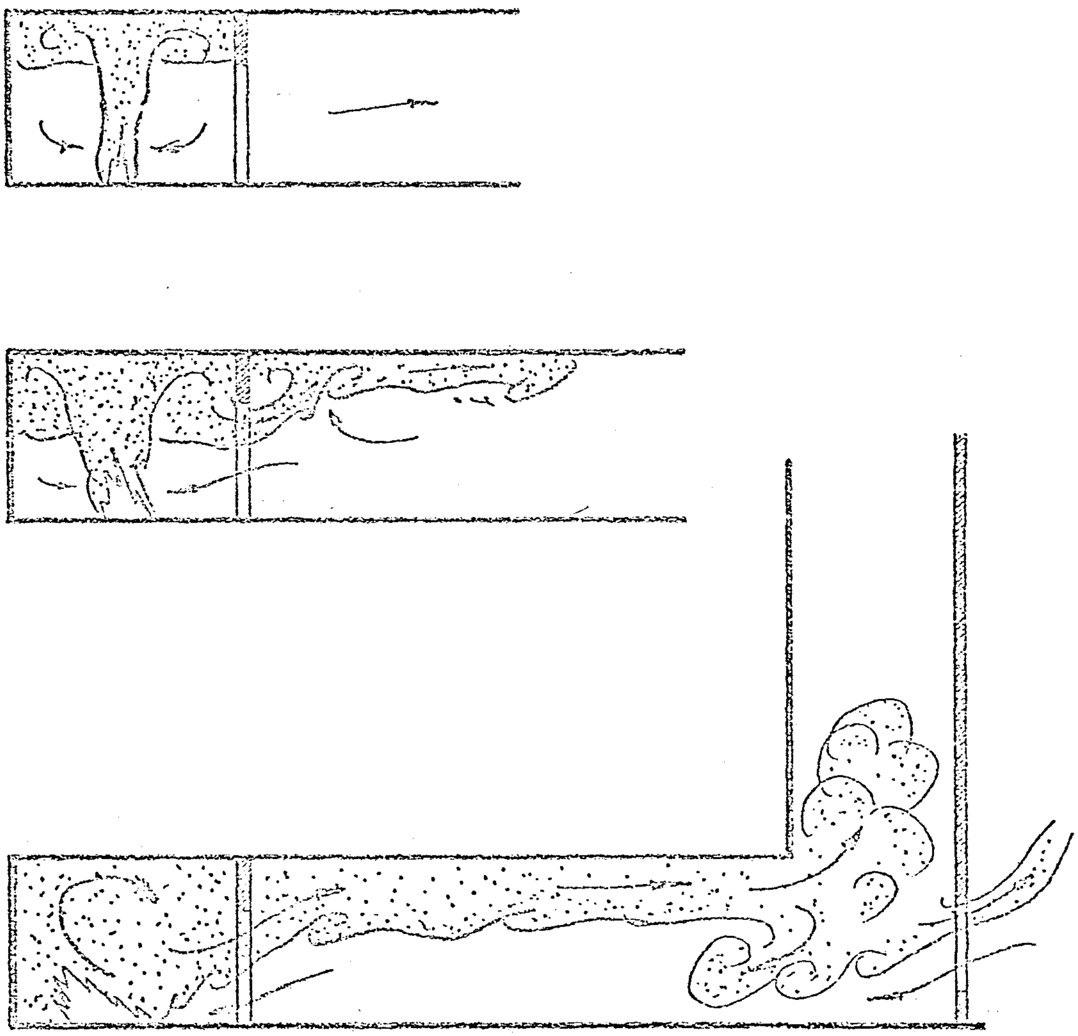


Figure 1. Sketches of the Development of a Room Fire.

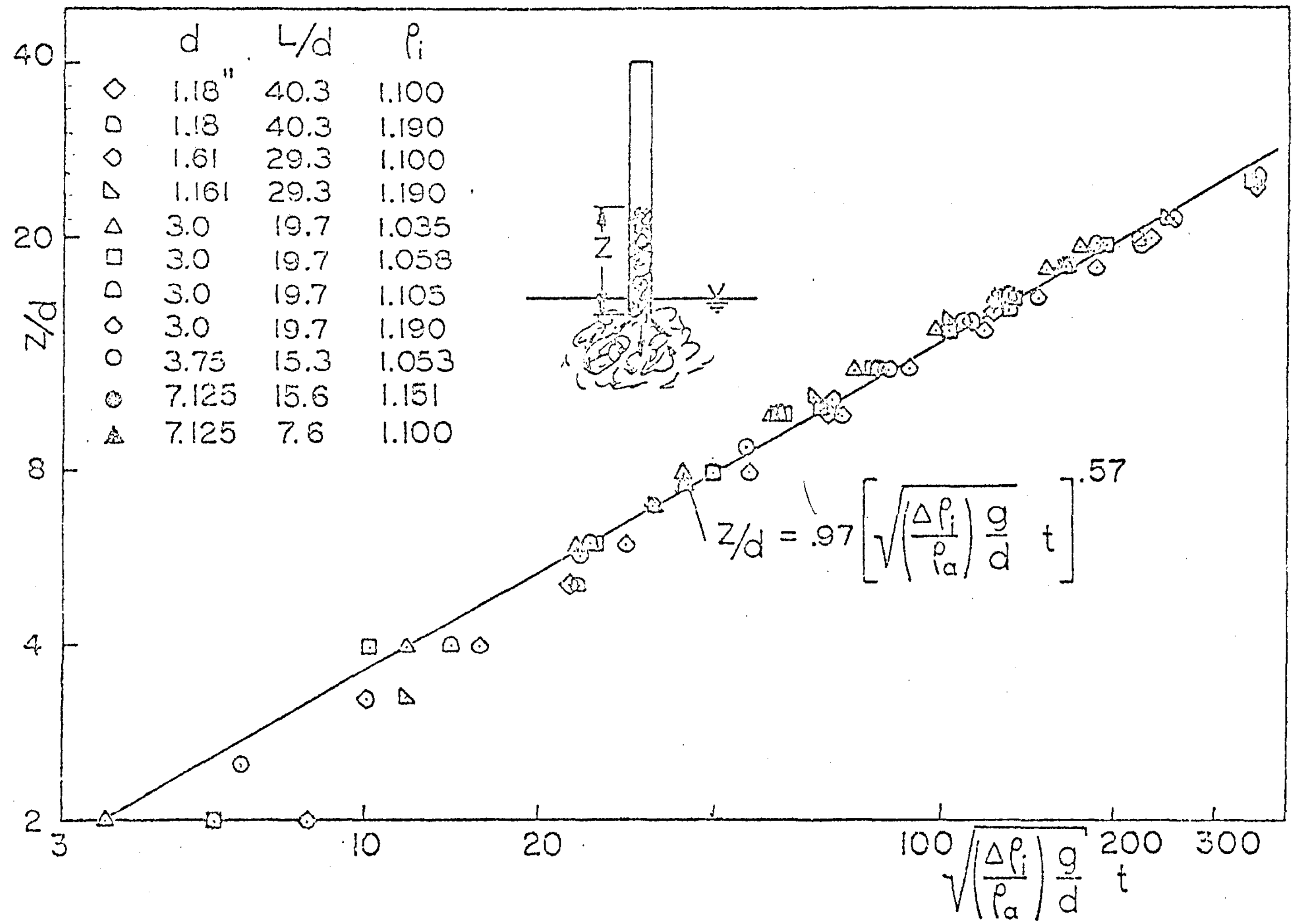


Figure 2. Propagation of the Initial Front in the Brine Solution Model.

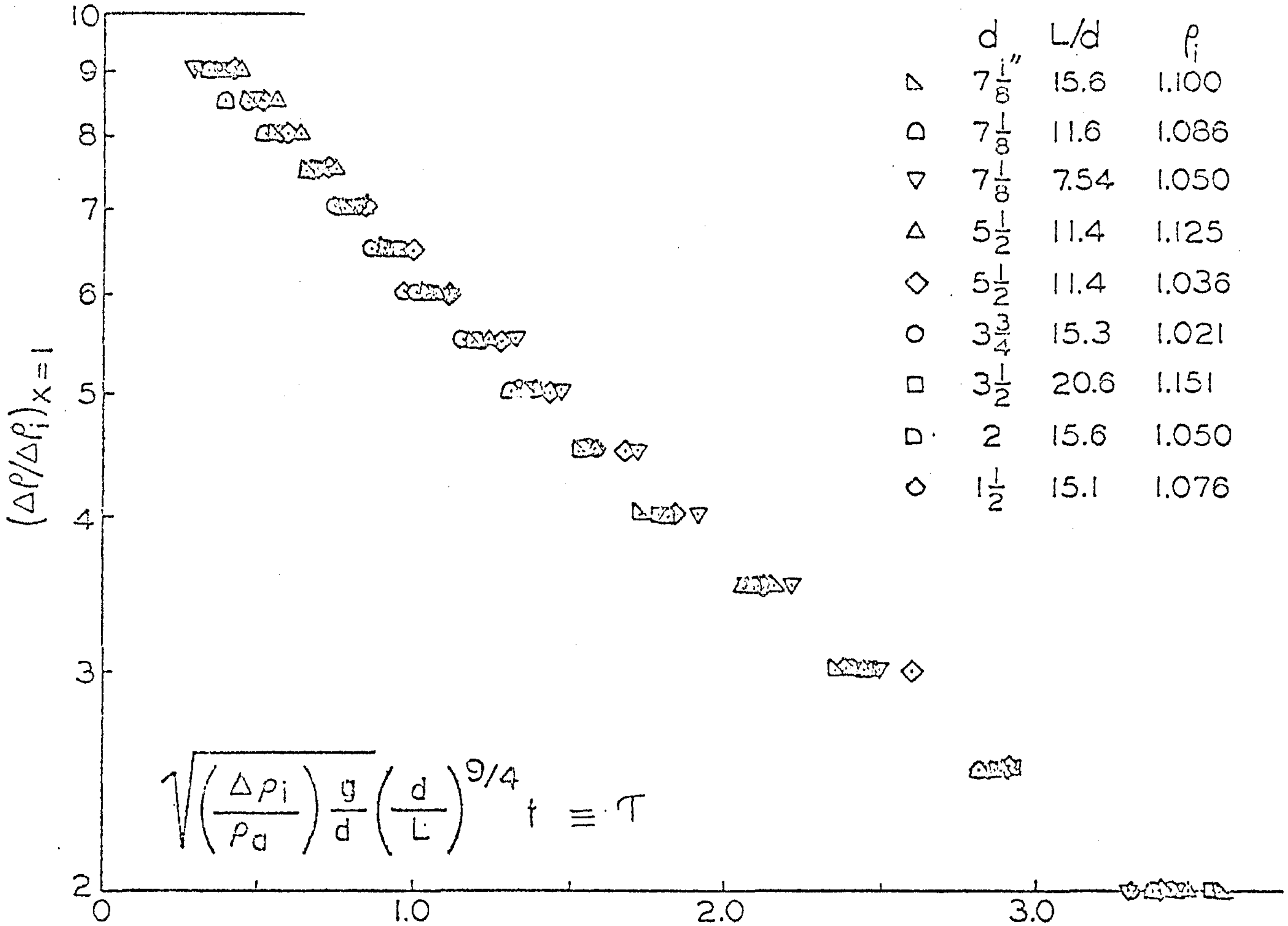


Figure 3. Decay of Density Difference with Time at the Top of the Shaft.

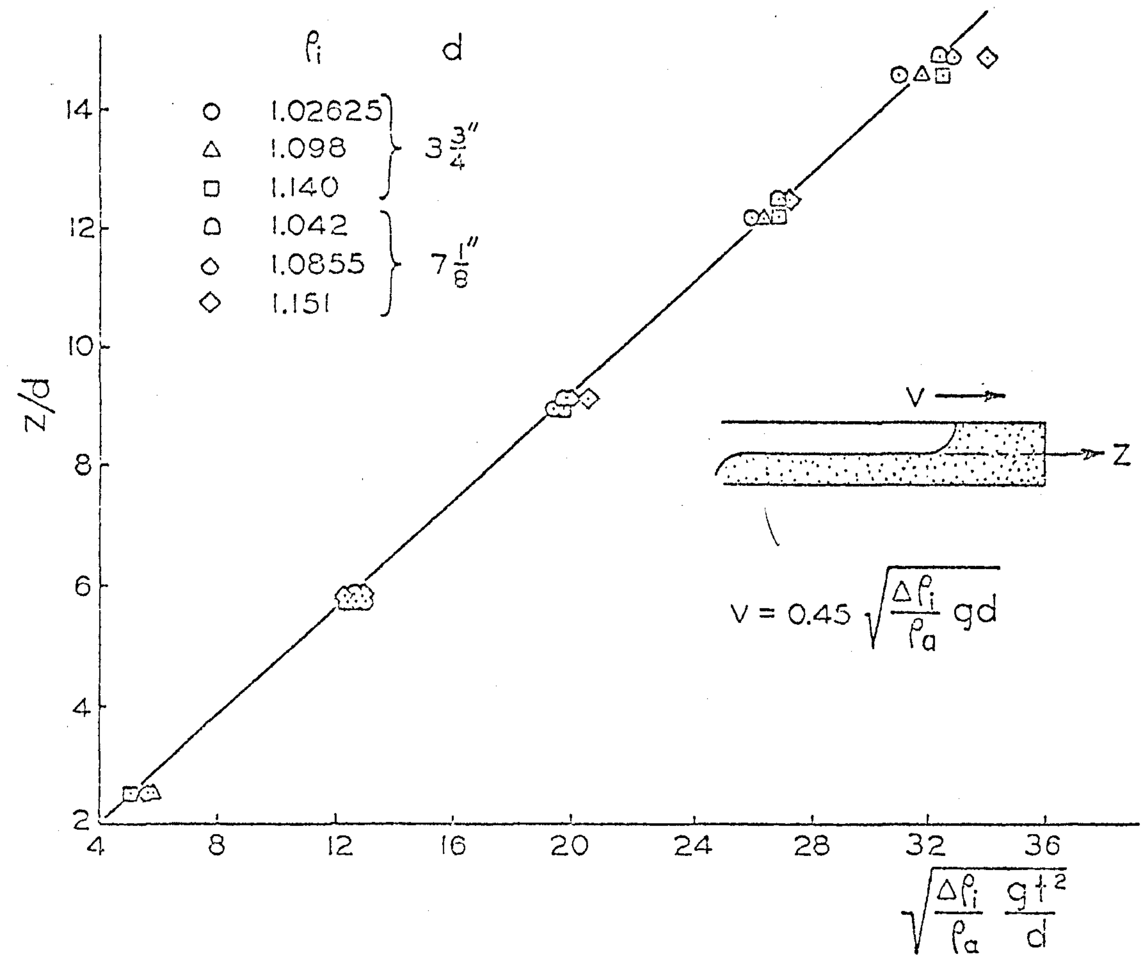
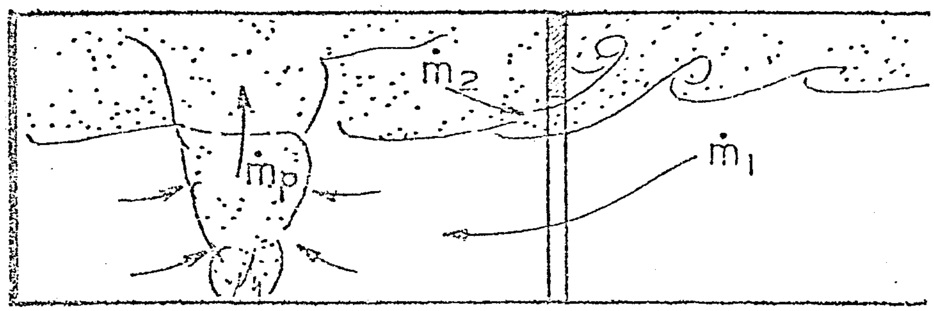


Figure 4. The Propagation of Density Currents in Corridors.



$\dot{m}_1 = \dot{m}$ $\dot{m}_2 = \dot{m}_p$

mass-averaged plume enthalpy = ceiling-layer enthalpy

Figure 5. Room Model.

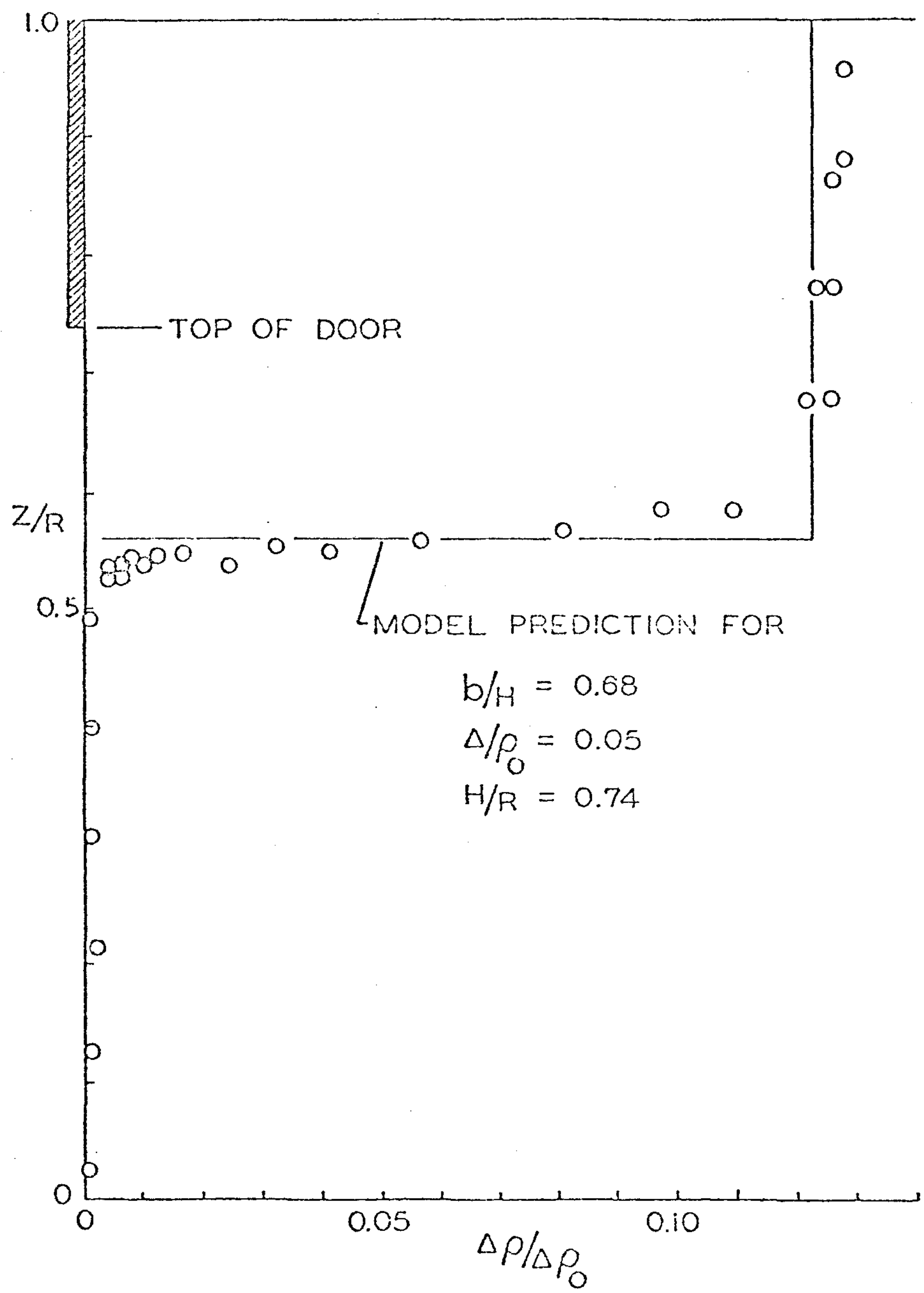


Figure 6. Density Profile in Room-Door Configuration with Axisymmetric Fire Plume.