CH-99-1-1

CFD Simulations of the Effects of HVAC-Induced Flows on Smoke Detector Response

John H. Klote, D.Sc., P.E.
Fellow ASHRAE

ABSTRACT

Rapid activation of fire protection systems in response to a growing fire is one of the important factors required to provide for life safety and property protection. Airflow due to the heating, ventilating, and air-conditioning (HVAC) system can significantly modify the flow of smoke along the ceiling and must be taken into consideration when a particular system is designed. At present, the standards used to guide the design of fire protection systems contain very little quantitative information concerning the impact of airflow produced by HVAC systems. This paper describes the results of a project that used computational fluid dynamics (CFD) to simulate smoke movement in response to HVAC-induced airflows. The HVAC simulations included ceiling-mounted slot diffusers, wall-mounted slot diffusers, high sidewall diffusers, and ceiling diffusers from which airflow drops to the floor, in combination with rectangular and slot returns. The CFD model was modified to calculate smoke detector activation times throughout the fire-driven flow field.

INTRODUCTION

Rapid activation of fire protection systems in response to a growing fire is one of the important factors required to provide for life safety and property protection. Airflow due to the heating, ventilating, and air-conditioning (HVAC) system can significantly modify the flow of smoke along the ceiling and must be taken into consideration when a particular system is designed. At present, the standards used to guide the design of systems contain very little quantitative information concerning the impact of airflow produced by HVAC systems.

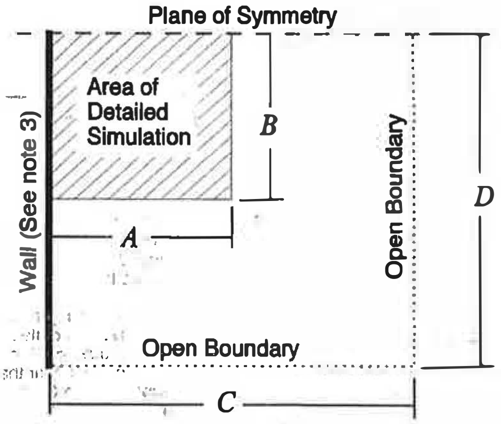
The International Fire Detection Research Project sponsored by the National Fire Protection Research Foundation (NFPRF) was established to provide quantitative information on the impact of beamed and sloped ceilings and HVAC flows on the distribution of heat and smoke during a fire. During the first year of the project (Forney et al. 1993), validation simulations were made to compare the results of numerical modeling with the experimental data of Heskestad and Delichatsios (1977), and additional simulations of smoke movement under level, beamed ceilings were made. In the second year, numerical simulations of smoke movement in response to sloped, beamed ceilings were made (Davis et al. 1994). The third and fourth years of the project consisted of the numerical simulations of smoke movement and smoke detector response to HVAC-induced flows. The third year simulations focused on slot diffusers with slot returns and rectangular returns (Klote 1997).

This paper describes the results of the fourth year simulations of this project and presents observations and conclusions based on both the third and fourth year effort. The fourth year simulations examined the effects of HVAC flows resulting from ceiling-mounted slot diffusers, wall-mounted slot diffusers, high sidewall diffusers, and ceiling diffusers from which airflow drops to the floor.

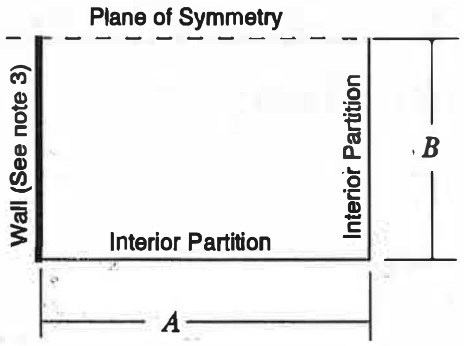
Because of unexpected year 3 results in the vicinity of the returns in open plan rooms, the ceiling returns in year 4 included simulation of flows in a plenum space (above the ceiling) in an effort to improve the simulation in the vicinity of the return. Detector activation was based on the simulated mass density approach developed in year 3. Simulations of smoke movement, including HVAC effects, were made for five enclosed rooms and two open plan rooms (dimensions listed in Table 1). Open plan rooms are ones where the floor

John H. Klote is a consultant in fire and smoke control at John H. Klote, Inc., McLean, Va. While working on this paper, Dr. Klote was a research engineer at the National Institute of Standards and Technology.

TABLE 1
Room Dimensions in Meters (Feet) for CFD Simulations



Open plan room (O)



Enclosed room (E)

Notes:
1. *H* is floor to ceiling height.
2. NA is not applicable.
3. Wall is an exterior wall for rooms O2, E3, and E4; it is interior partition for rooms O1, E1, E2 and E5.

Room Type	A	B	C	D	H
O1	4.6 (15)	4.6 (15)	9.2 (30)	9.2 (30)	2.74 (9.0)
O2	5.5 (18)	3.91 (13)	10.5 (34)	9.09 (30)	2.74 (9.0)
E1	4.6 (15)	4.6 (15)	N/A	N/A	2.74 (9.0)
E2	4.6 (15)	3.0 (9.8)	N/A	N/A	2.74 (9.0)
E3	4.6 (15)	1.71 (5.6)	N/A	N/A	2.74 (9.0)
E4	4.6 (15)	2.56 (8.7)	N/A	N/A	2.44 (8.0)
E5	6.0 (20)	2.3 (7.5)	N/A	N/A	2.44 (8.0)

area is very large as is the case for open plan offices, and enclosed rooms are ones that are relatively small with walls like those of private offices. For these rooms, the fire locations and the arrangements of HVAC supplies and returns are shown in Figure 1.

MODELING APPROACH

A commercially available computational fluid dynamics (CFD) model was used to perform the numerical simulations. This model (CFC 1995) is an upgrade of the one used for simulations in the second year of this project, and the only upgraded features relevant to the simulations of this study consist of simplified data input.

A detailed description of CFD modeling is beyond the scope of this report. The nonmathematical descriptions in the following sections are intended to provide an explanation of the assumptions of the simulations of this paper and to provide insight for those not familiar with the field. The user's manual (CFD 1995) provides the exact equations and mathematical definitions that apply to these simulations. For more information about CFD modeling, readers are referred to Abbott and Basco (1989), Anderson et al, (1984), Hirsch (1990), Hoffmann (1989), and Kumar (1983). For general information about fluid dynamics, readers are referred to Schetz (1993), Schlichting (1960), Sherman (1990), White (1974), and Yuan (1967).

CFD Concept

The CFD modeling consists of dividing the flow field into a collection of small rectangular cells and determining the flow at each cell by numerically solving the governing conservation equations of fluid dynamics. Boundary conditions are prescribed for walls, floor, ceiling, HVAC supplies, HVAC returns, openings to the outside, and planes of symmetry. For this project, all the CFD simulations were unsteady, using the calculated properties from one time step to calculate those of the next time step. At the start of a simulation, each cell can be set to zero flow conditions or conditions read into the computer from a previous simulation. Zero flow conditions consist of zero velocity and ambient pressure and temperature. To generate fire-induced flows, heat is released in several control volumes over time.

The governing equations of fluid dynamics describe the motion of fluid throughout the flow field. These equations are

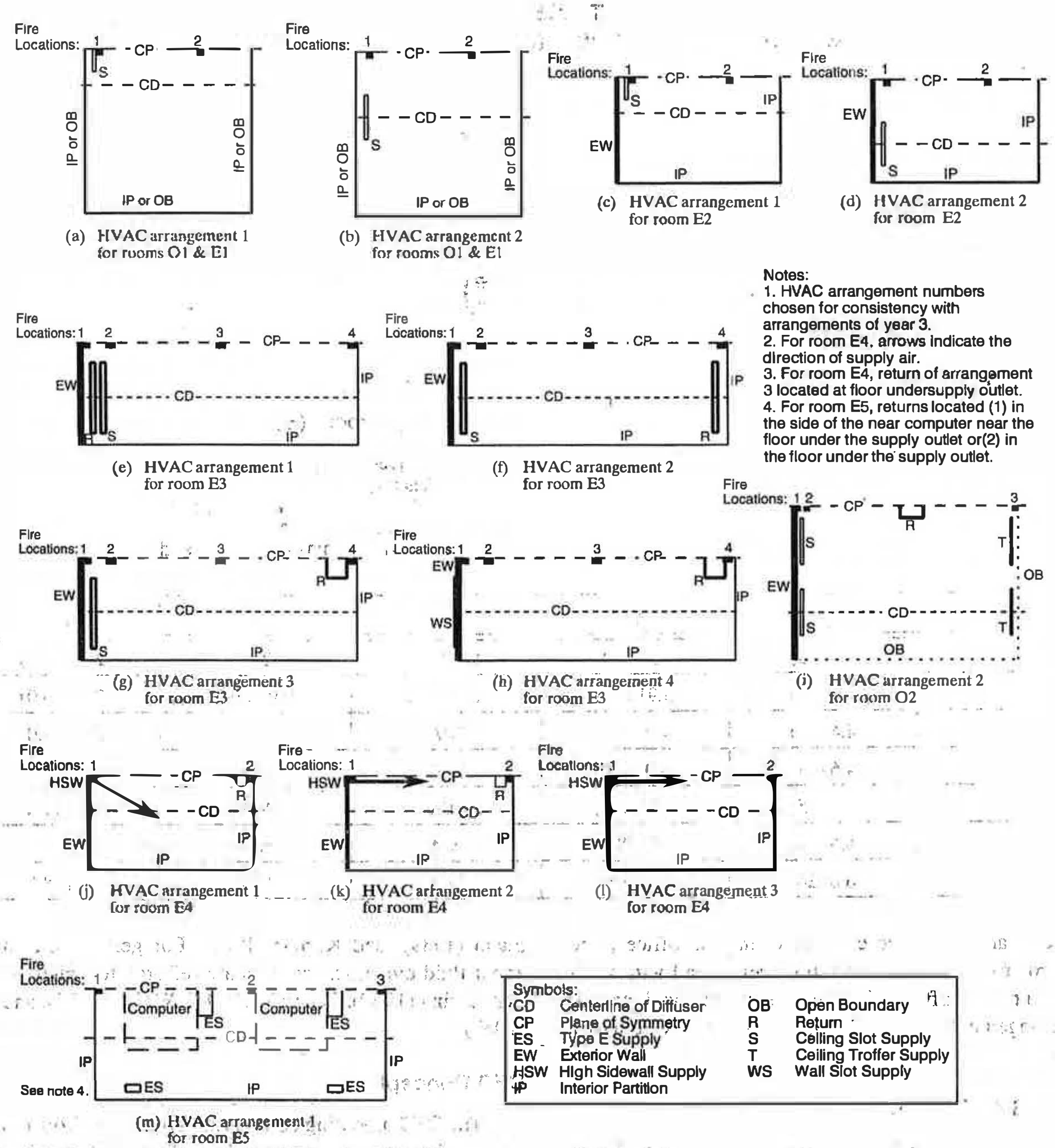


***Figure 1** Fire locations and HVAC arrangements.*

(1) conservation of mass, (2) conservation of momentum, and (3) conservation of energy. The mass conservation equation depends on the concept that matter is not created nor destroyed for the processes of interest in fluid dynamics. The momentum conservation or the Navier Stokes equations are equivalent to Newton's second law of motion. The energy conservation equation is equivalent to the first law of thermodynamics. In addition to these conservation equations, the ideal gas law relates density, pressure, and temperature. The conservation equations are expressed mathematically as a set of simultaneous, nonlinear, partial differential equations. This set of equations is solved numerically for each cell to simulate the motion of fluid. The following section provides specific information about the simulations of years 3 and 4, and further details are provided by Klote et al. (1996).

Specifics of CFD Simulations

Turbulence. The k-ε model developed by Launder and Spalding (1974) was used to account for the effects of turbulence on a scale smaller than that of the cells.

Heat Transfer. The solid surfaces (walls, floor, and ceiling) were considered to be adiabatic. Radiation effects were not included explicitly in the calculation except that only a fraction of the heat release rate was assumed to contribute to convective heating of smoke and air. The rest of the heat was considered to be radiated away. These assumptions were evaluated for year 2 of the project, and it was shown that these assumptions resulted in negligible variations in activation time.

Area Modeled and Gridding. A number of grid arrangements were needed to represent the seven room sizes and the combinations of HVAC supplies and returns. The total number of grid cells ranged from 11,000 to 37,000. As in the earlier years, the grids have smaller spacing at locations where more flow detail is needed: the fire, the slot diffuser, and the slot returns near the ceiling (Figure 2).

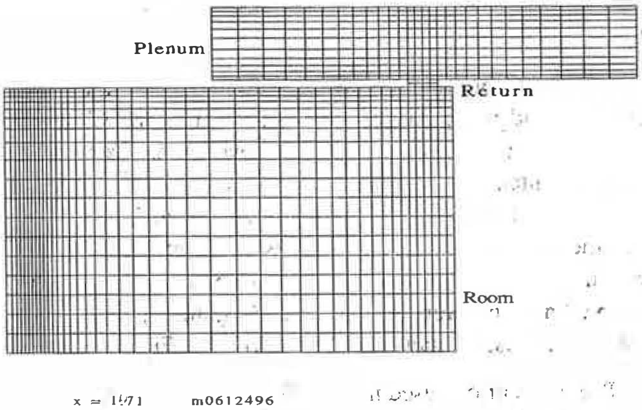


Figure 2 Grid typical of those used for simulations.

Initial Conditions. For these time-dependent analyses, the conditions (velocity, temperature, pressure, effective viscosity, kinetic energy of turbulence, and turbulent dissipation rate) at the start of each simulation must be defined for each cell. Because describing these initial conditions at every cell would be extremely intricate, the only practical way of determining them is by use of the CFD model. The initial conditions were obtained by CFD simulations without a fire but with the same gridding and boundary conditions as the simulations with fires.

Boundary Conditions. The following boundary conditions were used: (1) solid surface, (2) plane of symmetry, (3) velocity at an opening, and (4) pressure at an opening. The boundary conditions remained constant throughout each simulation. At the solid surfaces (walls, floors, and ceilings) the velocity was assumed to be zero (no slip condition). As with a solid surface, there is no flow through a symmetry boundary, but there can be flow at a symmetry boundary provided that the direction of such flow is in the plane of the boundary. Both velocity and pressure boundaries can be used where mass is to enter or leave the flow domain. Velocity boundaries are used to define the velocity entering or leaving the domain. For pressure boundaries, the pressure is defined by the user, and the CFD model calculates the velocity.

Fire. For all of the simulations of years 3 and 4, the heat release rate (HRR) of the fire was proportional to the square of the time from ignition. At ignition, the HRR was zero, and at 84.75 seconds after ignition, HRR was 100 kW. Such a fire is referred to as a medium growth t-squared fire (NFPA 1993). The fire was modeled by releasing energy over several grid cells. As in prior years, the number of cells occupied by the fire was varied during the simulation such that the heat release rate per unit volume would not exceed 2.6 MW/m^3 (0.070 Btu/s ft). The fire was situated at various locations on the plane of symmetry. To account for radiative losses from the fire to the walls and ceiling, the heat release prescribed for the fire was reduced by 35%.

Smoke Generation and Movement. As in year 3, the movement of smoke was simulated by a species mass fraction approach by using the scalar equation feature of the CFD model. It should be noted that the particulate motion simulated does not simulate smoke particle aging (particulate agglomeration and deposition).

HVAC SUPPLIES

The locations of the supply outlets are shown in Figure 1. These outlets had the following flow rates:

Slot	0.106 m^3/s (225 cfm)
Troffer	0.0236 m^3/s (50 cfm)
High sidewall	0.318 m^3/s (674 cfm)
Group E above computer	0.27 m^3/s (572 cfm)
Group E at room edge	0.045 m^3/s (95 cfm)

Group E outlets are the ones located in the ceiling from which air drops toward the floor.

HVAC RETURNS

A general HVAC design rule is that the location of a return has almost no effect on room air distribution, provided that the supply is not blowing directly into a return. At the beginning of year 3, it was erroneously thought that this rule could be extended to indicate that returns have an insignificant effect on detector activation. Because this rule is so strongly held, it was felt that any errors in CFD-simulated velocities in the vicinity of the return would not have a significant effect on detector activation. For this reason, year 3 returns were simulated as pressure boundaries for enclosed rooms and as velocity boundaries for open-plan rooms. However, year 3 results indicated a strong influence of returns on detector activation. Thus, more detailed treatment of returns was used in year 4 to evaluate the unexpected results.

As already stated, year 4 returns included simulation of a portion of the plenum space above the ceiling (Figure 2). For enclosed rooms, the sides of the plenum section (other than the sides that are part of exterior walls or part of the plane of symmetry) are pressure boundaries. For the initial condition simulation, these boundary conditions resulted in the flow out of the return equaling the supply flow. For the fire simulations, the CFD software calculated the return flow, accounting for the effects of expansion of gases inside the room due to the fire.

In order to simulate the complicated flows in and out of the large open boundaries of the open plan room (Figure 1a, 1b, and 1i), pressure boundaries were needed. However, using pressure boundaries for these large open areas resulted in difficulties concerning use of pressure boundaries for the sides of the plenum section. The pressures assigned to the large open areas and to the plenum sides would need to be such that the desired flows occurred at the HVAC returns. The CFD model could be used to determine such pressures at these boundaries without a fire. However, fire-induced pressure changes near the plenum sides could result in significant error in the return airflow.

For the open plan simulations, all the sides of the plenum were walls (or part of the plane of symmetry) except for one side that was a velocity boundary. The velocity boundary was chosen at the plenum side farthest from the exterior wall, resulting in a plenum flow direction toward the building core. The value assigned to the velocity boundary was such that the average velocity at the return was 2 m/s (400 fpm) for runs 43, 44, 47, and 48. This is the return velocity used for the open plan simulations of year 3. The resulting return flow is approximately equal to the air supplied to the slots and troffers in the total space simulated (including a slot and a troffer outside the area of detailed simulation).

To examine the effect of a greater return velocity, runs 45 and 46 had average return velocities of 3 m/s (600 fpm). This return, with its greater flow, can be thought of as serving an occupied building space larger than the total space simulated. For the steady flow conditions, this results in a slight net flow into the space simulated at the open boundaries.

SIMULATIONS

Year 4 simulations are divided into the following groups: transition study, room with ceiling slot supply, open plan room with ceiling slot supply, room with wall slot supply, room with high sidewall supply, and computer room. The simulations are summarized in Table 2. As previously stated, Table 1 lists the room dimensions and Figure 1 shows the fire locations and the arrangements of HVAC supplies and returns.

The results of the simulations are shown in Figures 2 through 8. Each figure shows activation times at 0.02 m (0.79 in.) and 0.05 m (2.0 in.) below the ceiling with the intent of providing insight into the activation times of low profile and normal detectors. Differences between simulated results at these levels must be interpreted with caution because the levels are only about one cell apart and, thus, the differences in simulated values are highly dependent on the empirical turbulence model.

Transition Simulations

The transition simulations consisted of runs 32 through 36. The unrealized purpose of the these simulations was to develop information about the transition between the two very different patterns of nonactivation observed in year 3 between enclosed rooms and open plan rooms. In year 3, the nonactivation distances in front of the slot supply of the open plan rooms ranged from 1.1 m to 3.3 m (3.6 ft to 10.8 ft), and those in front of the slot supply of the enclosed rooms ranged from 0 m to 1 m (0 ft to 3.3 ft).

Transition simulations consisted of arrangements of a slot diffuser in open plan rooms and enclosed rooms with open doors, as shown in Figure 1a and 1b. These arrangements were selected to allow for comparison between the different rooms and were not representative of realistic HVAC conditions. Further research is needed to understand this transition.

Room with Ceiling Slot Supply

Runs 37 through 42 are in an enclosed room with ceiling slot supply and plenum return. These are reruns of year 3 simulations except that in year 3, the returns were pressure boundaries. The intent of these simulations was to determine the extent to which returns influence activation time. The activation time is almost the same for these simulations (runs 37 through 42) as for the similar runs without plenum returns. For example, the activation times of run 3 (Figure 11; Klote et al. 1996) are almost the same as those of run 37 (Figure 3).

The conventions used in these figures are the same as for the other figures of activation time in this report and the report of year 3. For example, Figure 3a shows the activation time at 0.02 m (0.79 in.) below the ceiling. Activation directly above the fire took 20 seconds or less (second darkest area on the key in Figure 3a). The white space in Figure 3a is where activation takes 80 seconds or more (it should be noted that the irregular boundary on Figure 3 between the white region [>80 seconds] and the next region [65 to 80 seconds] is an artifact of graphics. Similar irregular boundaries occur on most of the other activation time plots in this paper). The white spaces where activation takes longer than 80 seconds will be referred to as areas of nonactivation. The spaces where activation is 80 seconds or less will be referred to as the areas of activation. The range from 20 to 80 seconds was selected to simplify the scales, and these figures show the same trends as ones that go up to the full 84.75 seconds of simulation.

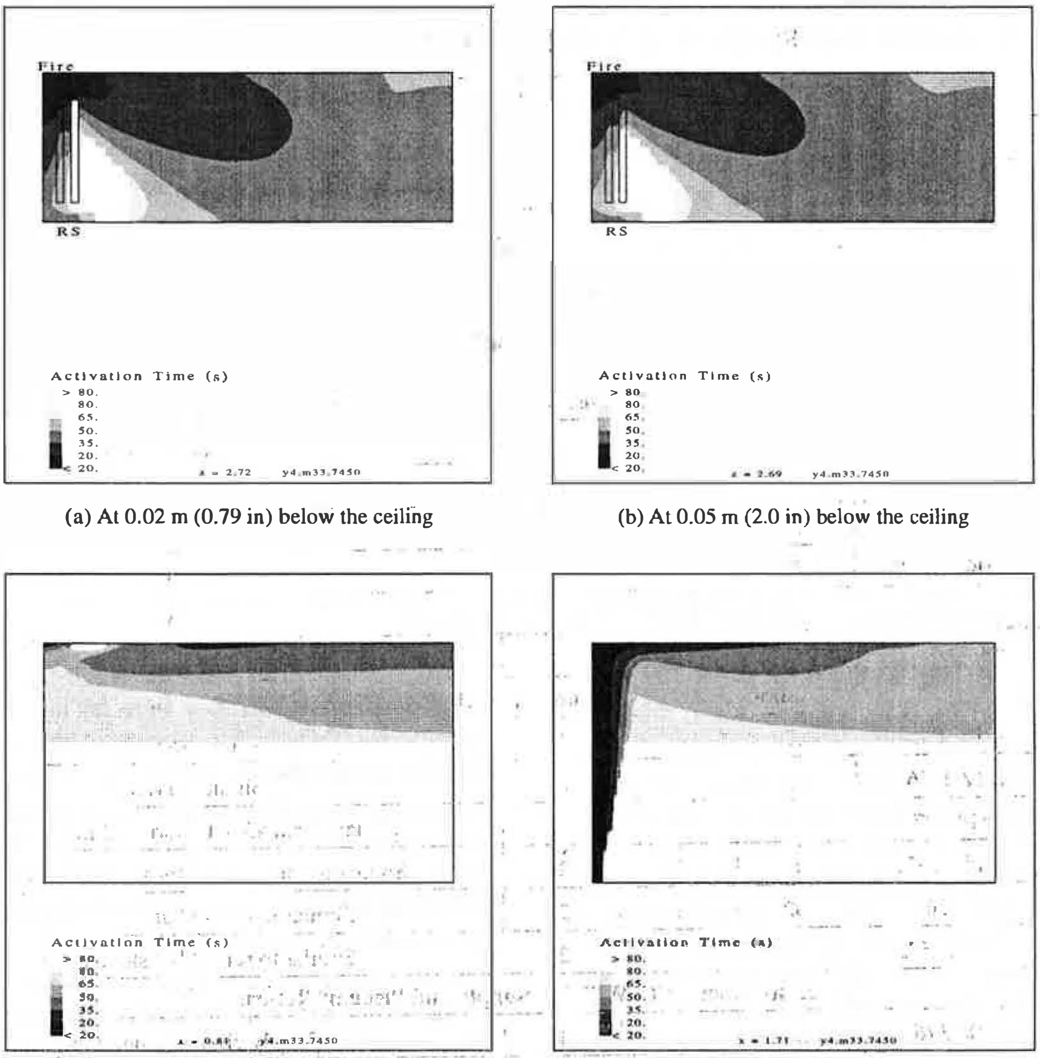
Open Plan Room with Ceiling Slot Supply

Six simulations of detector activation were made for an open plan room with ceiling slot supply, troffer supply, and plenum return (runs 43 through 48). The first two simulations were reruns of year 3 simulations except that in year 3 the returns were velocity boundaries. The intent of these simulations was to determine if nonactivation areas adjacent to the return in year 3 simulations would occur with plenum returns. Other simulations examined the effect of increased return air velocity (runs 45 and 46) and of dumping supply air toward the floor (runs 47 and 48).

TABLE 2
Summary of Smoke Movement Simulations

Run	ID Number	Room[1]	Fire[2]	HVAC[2]	Comments
Transition Study					
32	m19.5196	O1	2	1	Open plan room
33	m23.1028	E1	2	1	Enclosed room with open door
34	m25.13988	E2	2	1	Enclosed room with open door
35	m20.26725	O1	1	2	Open plan room
36	m27.21477	E2	1	2	Enclosed room with open door
Enclosed Room with Ceiling Slot Supply and Plenum[3] Return					
37	m33.7450	E3	1	1	Similar to run 3
38	m35.9175	E3	4	2	Similar to run 10
39	m37.21853	E3	1	3	Similar to run 11
40	m04a.22733	E3	2	3	Similar to run 12
41	m39.25121	E3	3	3	Similar to run 13
42	m06.12496	E3	4	3	Similar to run 11
Open Plan Room with Ceiling Slot Supply, Troffer Supply, and Plenum[3] Return					
43	m11.13354	O2	1	2	Similar to run 26
44	m12.26160	O2	3	2	Similar to run 28
45	m14.4275	O2	1	2	Similar to run 43 but return at 3 m/s (600 fpm)
46	m15.10073	O2	3	2	Similar to run 44 but return at 3 m/s (600 fpm)
47	m41.3667	O2	1	2	Similar to run 43 but slot dumps air
48	m42.15672	O2	3	2	Similar to run 44 but slot dumps air
Enclosed Room with Wall Slot Supply and Plenum[3] Return					
49	m08.3767	E3	1	4	Supply discharge horizontal
50	m09.22437	E3	4	4	Supply discharge horizontal
51	m08a.27793	E3	1	4	Supply discharge 20° upward
52	m09a.16316	E3	4	4	Supply discharge 20° upward
Enclosed Room with High Sidewall Supply					
53	m29.16914	E4	1	1	Supply discharge diagonally and 20° upward; plenum[3] return
54	m30.18824	E4	2	1	Supply discharge diagonally and 20° upward; plenum[3] return
55	m44.11711	E4	2	2	Supply discharge across room and 20° upward; plenum[3] return
56	m46.6981	E4	2	3	Supply discharge across room and 20° upward; return at floor
Enclosed Computer Room with Group E[4] Supply					
57	m48.11682	E5	1	1	Supply outlets dump air toward floor
58	m49.25577	E5	2	1	Supply outlets dump air toward floor
59	m50.5402	E5	3	1	Supply outlets dump air toward floor

[1] See Table 1 for room dimensions.
[2] See Figures 3 through 8 for fire locations and HVAC arrangements.
[3] Simulation of ceiling returns includes a section of plenum 0.75 m (2.46 ft) in depth and extending 2 m (6.56 ft) in all horizontal directions from the return except that extensions do not go beyond the plane of symmetry or the exterior wall.
[4] Group E supply outlets are mounted in the ceiling, and air flows vertically from them toward the floor.



(a) At 0.02 m (0.79 in) below the ceiling

(b) At 0.05 m (2.0 in) below the ceiling

(c) Plane 1 m (3.3 ft) from plane of symmetry

(d) Plane of symmetry

Figure 3 *Simulated activation time in room E3 from run 37. (**Note:** See Figure 1 for the location of the plane of symmetry.)*

Room with Wall Slot Supply

All the previous simulations have had ceiling-mounted slot supplies or ceiling-mounted troffer supplies. The effect of flow from a wall-mounted slot supply on activation was simulated in an enclosed room with a plenum return (runs 49 through 52). The tops of these slots were 0.15 m (0.49 ft) below the ceiling on the exterior wall. For the first two of these simulations, the discharge velocity from the slot was horizontal. The direction of the discharge velocity of many slot diffusers can be adjusted, so the other two simulations in this group were done with a discharge velocity at a 20° upward angle.

Room with High Sidewall Supply

Runs 53 through 56 focused on the effect of high sidewall supply diffusers in an enclosed room. This diffuser was 0.6 m (1.97 ft) by 0.259 m (0.85 ft) high with the top located 0.15 m (0.49 ft) below the ceiling on the exterior wall. The first simulation examined the effect of a discharge velocity at an angle of 30° to the side and an angle of 20° upward. The other simulations in this group examined discharge velocities aimed straight across the room with a 20° upward angle. The first three simulations had rectangular ceiling returns that were simulated with plenums, and the fourth high sidewall simulation had a floor return located under the supply outlet.

Computer Room with Downward Airflow

Simulations were conducted for an enclosed computer room with group E supply outlets (runs 57 through 59). Some computer manufacturers make equipment intended for installation in rooms where the computer cooling air is supplied to the room through outlets in the ceiling. Figure 1m shows that the room has two computers with two ceiling outlets for computer cooling air and two smaller outlets for space cooling air.

TABLE 3
Summary of Nonactivation Distances at 0.05 m (2.0 in.) Below the Ceiling and Activation Layer Depths

Diffuser Type	Room[1]	No. of Runs[2]	Maximum Nonactivation Distance				Activation Depths Under Ceiling[5]			
			Diffuser Front[3]		Diffuser Side[4]		Minimum[6]		Maximum	
			m	ft	m	ft	m	ft	m	ft
Ceiling Slot[7]	O	17	2.0	6.6	0.5	1.6	0.0	0.0	0.4	1.3
Light Troffer	O	15	1.0	3.3	0.6	2.0	0.0	0.0	0.4	1.3
Ceiling Slot	E	29	1.0	3.3	0.2	0.7	0.0	0.0	2.0	6.6
Wall Slot	E	4	0.5	1.6	0.1	0.3	0.0	0.0	1.3	4.3
High Sidewall	E	4	4.1	13.5	0.3	1.0	0.0	0.0	1.5	4.9
Group E[8]	E	3	N/A	N/A	0.1	0.1	0.7	2.3	1.5	4.9

[1] O indicates open room, and E indicates enclosed room.
[2] A total of 57 runs were made that simulated HVAC flows, and some of the runs above are included twice because they included two types of diffusers.
[3] Values listed are the largest distance in front of the diffuser and perpendicular to its length. The diffuser front is taken to be the direction of flow.
[4] Values listed are the largest on a side of the diffuser.
[5] Activation depths located on the CD plane (Figure 1).
[6] Activation depths of 0.0 were simulated in front of the supply diffusers, except for run 31 where it was located at the edge of the area of detailed simulation.
[7] The areas of nonactivation are so large because the areas in front of the diffuser joins that around the return.
[8] Group E diffusers dump air toward the floor, and so the idea of a nonactivation distance in front of the diffuser is not applicable (N/A).

OBSERVATIONS

1. *Low Profile and Normal Detectors*: In most of the simulations, the activation areas at 0.05 (2.0 in.) below the ceiling are very similar to those at 0.02m (0.79 in.) below the ceiling (for examples see Figures 3 through 8). However, in some simulations, the nonactivation areas were somewhat larger at 0.05 (2.0 in.) below the ceiling than they were at 0.02 m (0.79 in.) below the ceiling. This trend toward similar activation areas at these elevations was observed for simulations with ceiling-mounted slots, light troffers, wall slots, high sidewalls, and group E outlets. This indicates that for the simulations with forced ventilation of this project, the activation time of low profile detectors can be the same or slightly faster than that of normal detectors.

2. *Fire Below Return in Enclosed Room*: When a fire is located below or nearly below a return in an enclosed room simulation, there is a tendency toward
   a. hot plume gases entering the return,
   b. decreased depth of activation in the room, and
   c. increased nonactivation distance in front of the slot diffuser.

3. *Nonactivation on Diffuser Side*: Many of the enclosed room and open plan simulations resulted in areas of nonactivation to the side of the diffuser, as indicated in Table 3. For the enclosed room simulations, many of these areas of nonactivation reached the wall of the room about 0.2 m (0.7 ft) from the diffuser. For open plan simulations, many of these areas of nonactivation extended beyond the area of detailed simulation. For examples of nonactivation on the side of the diffuser, see Figures 3, 4, 5, and 6.

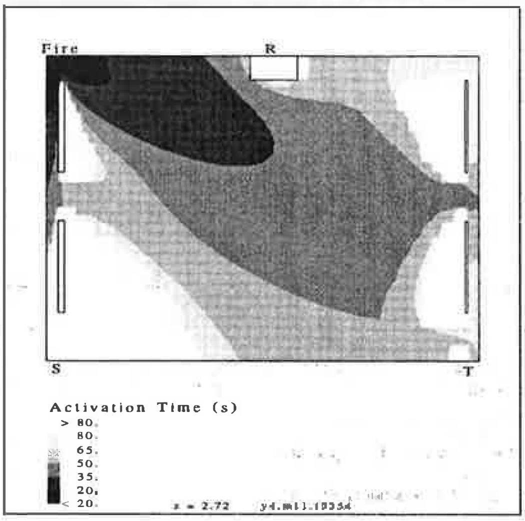
4. *Large Nonactivation Areas in Open Plan Rooms*: The nonactivation areas were much larger for the open floor plan simulations than they were for the enclosed room simulations (Table 3). For examples of large areas of nonactivation in open plan rooms, see Figures 4, 5, and 6.

5. *Activation Depth Small in Open Plan Rooms*: The activation depths were much smaller for the open floor plan simulations than they were for the enclosed room simulations (Table 3). For examples of small activation depths in open plan rooms, see Figures 4, 5, and 6.

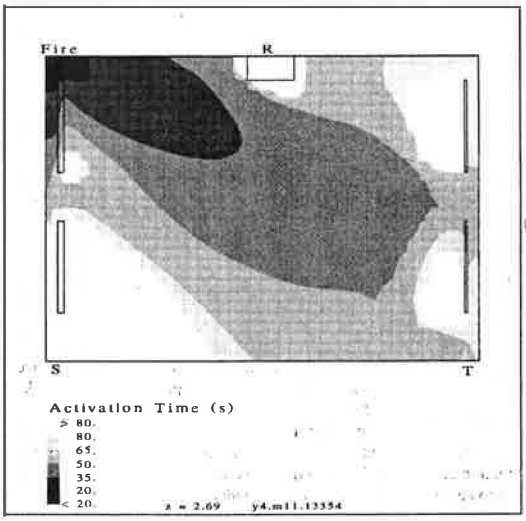
6. *Nonactivation Areas at Returns in Open Plan Rooms*: Areas of nonactivation occurred at the returns in some of the open plan room simulations. This nonactivation was unexpected in year 3, but year 4 analysis with detailed plenum return simulations resulted in similar nonactivation adjacent to the return. For examples of nonactivation areas at returns in open plan rooms, see Figures 4, 5, and 6. Increasing the return velocity from 2 m/s (400 fpm) to 3 m/s (600 fpm) resulted in only a slight increase in nonactivation area around the return (Figures 5 and 6).

7. *Nonactivation Dependent on Discharge Angle*: For the wall slot simulations, the nonactivation distance in front of the supply was dependent on the upward discharge angle of the air at the diffuser. It is expected that the discharge angle would have a similar impact for high sidewall diffusers. Because the discharge angle can be field adjusted, the diffuser adjustment that can be made by the building operating personnel can have an impact on nonactivation near diffusers.

8. *High Sidewall Nonactivation*: The high sidewall diffuser simulations resulted in the largest nonactivation distances observed in enclosed rooms in this study (Table 3, Figure 7). However, considering the small number of these simulations, further study is needed before conclusions can be reached concerning nonactivation and sidewall diffusers.

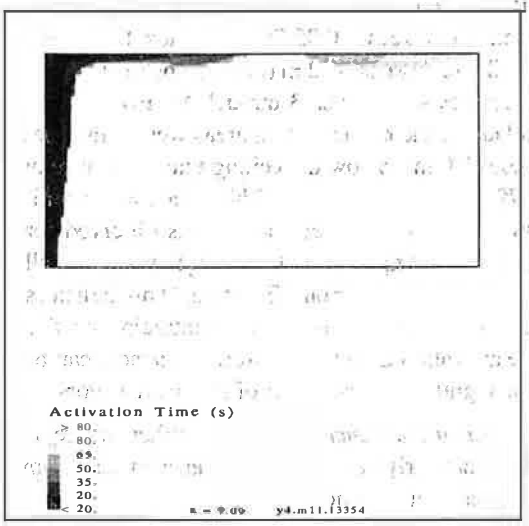


(a) At 0.02 m (0.79 in) below the ceiling



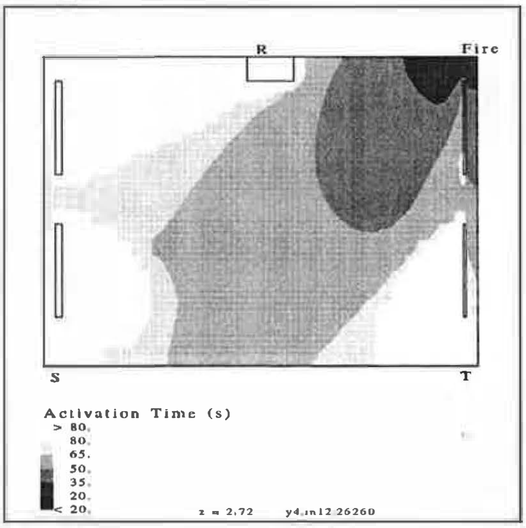
(b) At 0.05 m (2.0 in) below the ceiling

(c) Plane 1 m (3.3 ft) from plane of symmetry

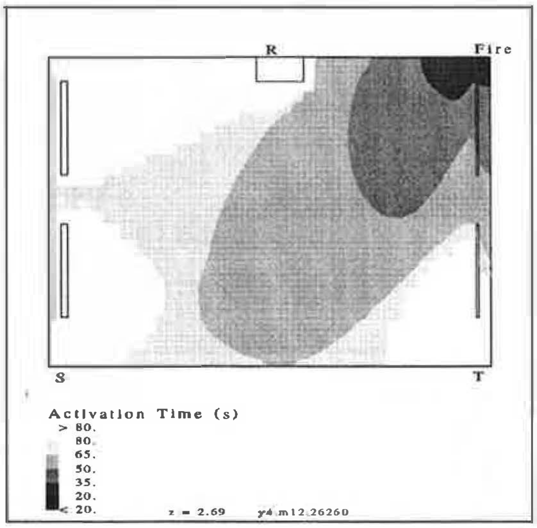


(d) Plane of symmetry

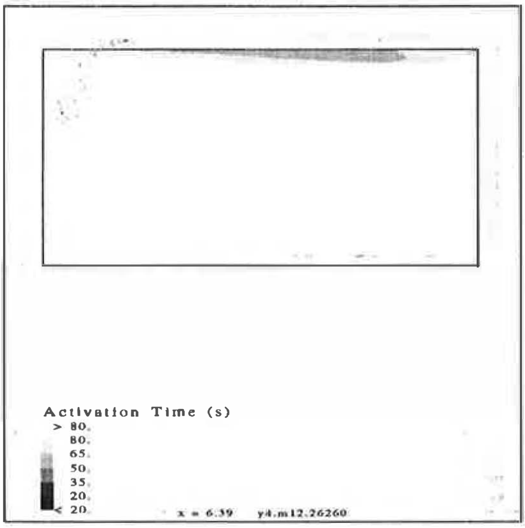
Figure 4 *Simulated activation time in room O2 from run 43. (Note: See Figure 1 for the location of the plane of symmetry.)*



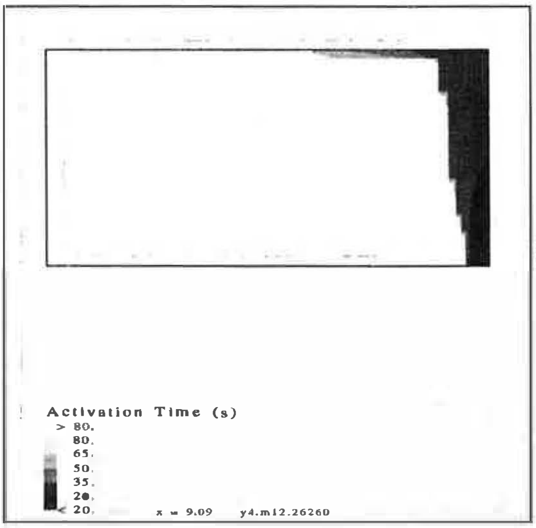
(a) At 0.02 m (0.79 in) below the ceiling



(b) At 0.05 m (2.0 in) below the ceiling

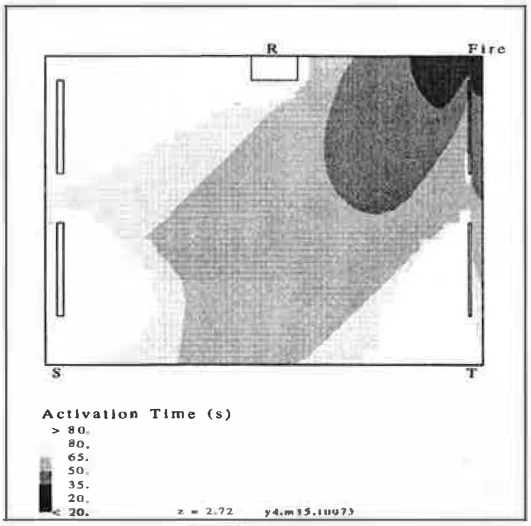


(c) Plane 1 m (3.3 ft) from plane of symmetry

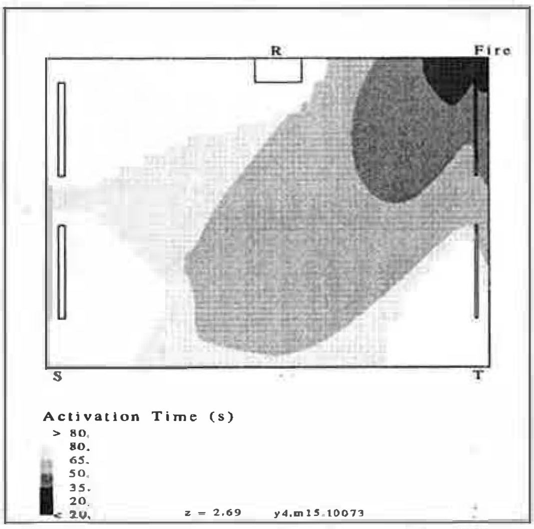


(d) Plane of symmetry

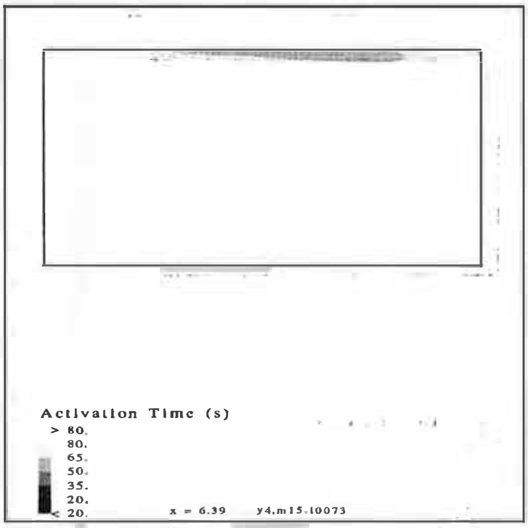
Figure 5 *Simulated activation time in room O2 from run 44. (Note: See Figure 1 for the location of the plane of symmetry.)*



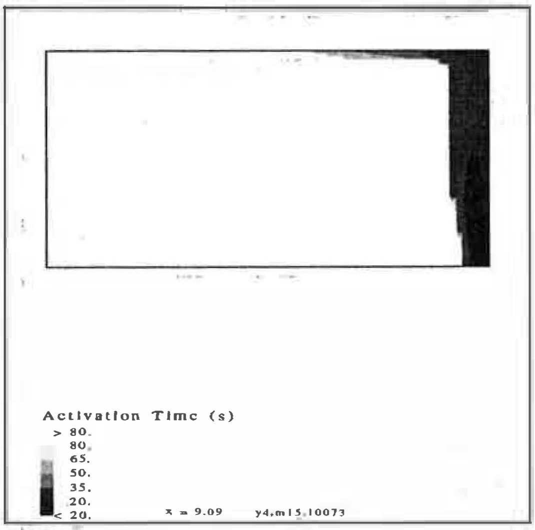
(a) At 0.02 m (0.79 in) below the ceiling



(b) At 0.05 m (2.0 in) below the ceiling

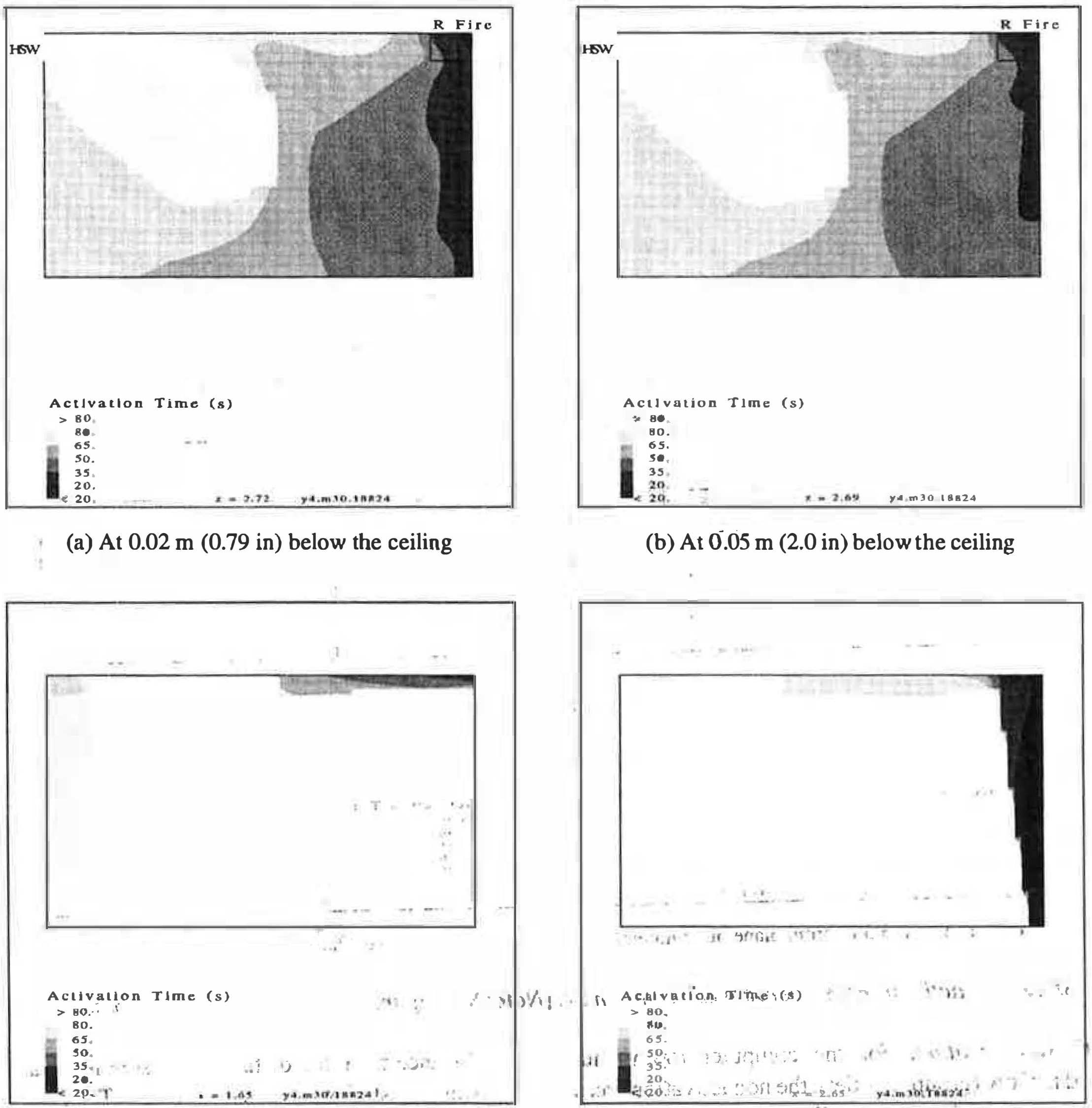


(c) Plane 1 m (3.3 ft) from plane of symmetry



(d) Plane of symmetry

Figure 6 *Simulated activation time in room O2 from run 46. (Note: See Figure 1 for the location of the plane of symmetry.)*



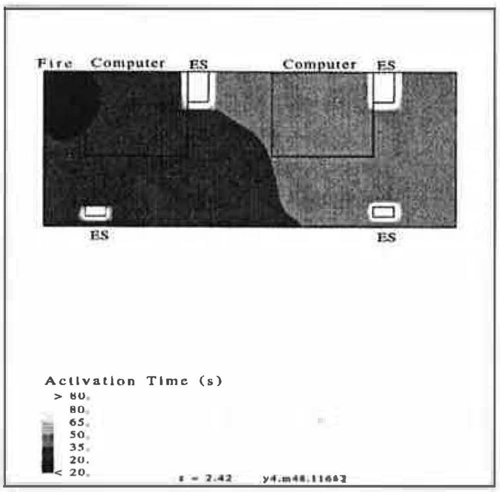
(a) At 0.02 m (0.79 in) below the ceiling

(b) At 0.05 m (2.0 in) below the ceiling

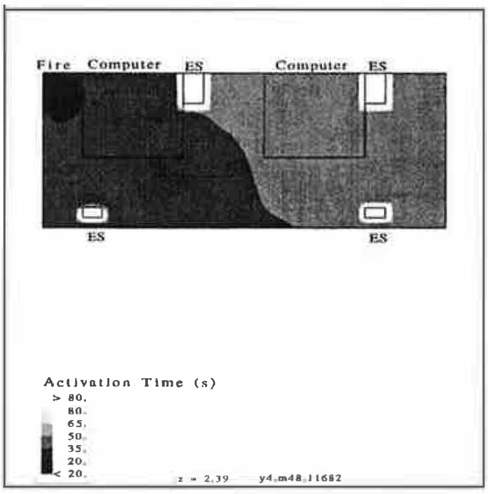
(c) Plane 1 m (3.3 ft) from plane of symmetry

(d) Plane of symmetry

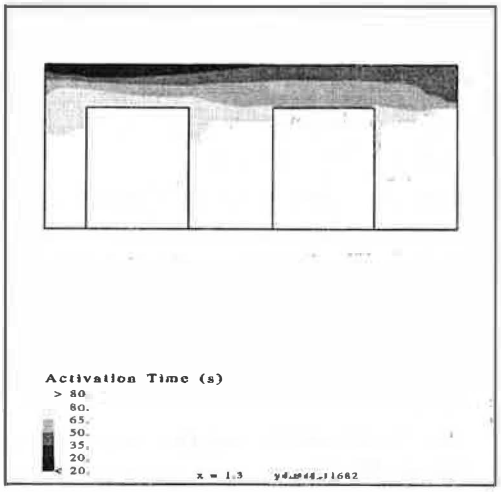
Figure 7 *Simulated activation time in room E4 from run 54. (Note: See Figure 1 for the location of the plane of symmetry.)*



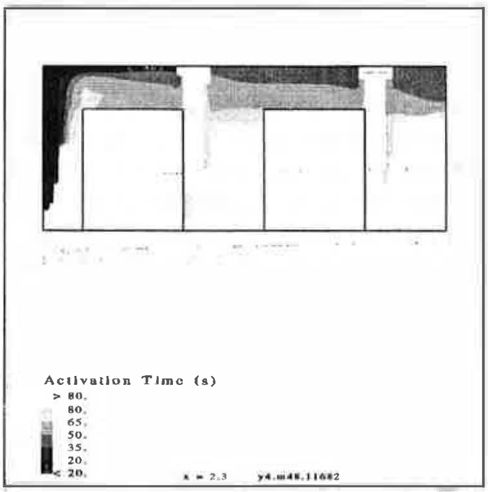
(a) At 0.02 m (0.79 in) below the ceiling



(b) At 0.05 m (2.0 in) below the ceiling



(c) Plane 1 m (3.3 ft) from plane of symmetry



(d) Plane of symmetry

Figure 8 *Simulated activation time in room E5 from run 57. (Note: See Figure 1 for the location of the plane of symmetry.)*

9. *Group E Nonactivation*: For the computer room with downward airflow (group E outlet), the nonactivation areas were located under the supply grilles and extended about 0.1 m (0.3 ft) to the sides of the grilles (Figure 8). The nonactivation distances were also about the same size for the open plan room simulation with the slot diffuser dumping air downward.

PHYSICAL INTERPRETATION OF NONACTIVATION

The following discussion is intended to provide some insight into the reasons for the major nonactivation areas observed in the simulations of this study.

- *Nonactivation in Front of Diffusers*: As expected, the HVAC supply jet caused nonactivation areas in front of the diffusers. It is well known that such jets entrain ambient air so that their velocity decreases with distance from the diffuser and their mass flow rate increases with distance from the diffuser. The supply air at the diffuser was "fresh," without any smoke. These jets were surrounded by smoke so that they entrained soot particulates, and the particulate concentration of the jet increased with distance from the diffuser. As the fire grew, the concentrations surrounding the jet increased, and the concentrations within the jet increased. This explains why activation takes longer in the HVAC jet front of the diffuser.
- *Nonactivation at Diffuser Sides*: Significant nonactivation resulted at a side of a diffuser when a fire was located at the other side of the diffuser (for example, Figure 3). A plume of fire gases rises above the fire to the ceiling, and the fire gases spread as a jet under the ceiling. This flow of hot gases under the ceiling is well known and is referred to as a ceiling jet. When unobstructed, the ceiling jet flows radially from where the plume impacts the ceiling. When the ceiling jet impacts