COMPUTING THE EFFECT OF SPRINKLER SPRAYS ON FIRE INDUCED GAS FLOW

by

Glenn P. Forney,
and Kevin B. McGrattan
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

International Conference on Fire Research and Engineering, September 10-15, 1995. Orlando, FL. Proceedings. Sponsored by National Institute of Standards and Technology (NIST) and Society of Fire Protection Engineers (SFPE). D. Peter Lund and Elizabeth A. Angell, Editors. Society of Fire Protection Engineers, Boston, MA, 1995.

Computing the Effect of Sprinkler Sprays on Fire Induced Gas Flow

Glenn P. Forney and Kevin B. McGrattan
National Institute of Standards and Technology
Gaithersburg, Maryland 20899

1 Introduction

Over the past twenty years there has been much debate concerning the interaction of sprinklers and draft curtains in large storage facilities. At issue is whether or not the two fire protection systems are mutually beneficial. Draft curtains inhibit the spread of hot gases near the ceiling, In some cases this may accelerate the activation of sprinklers and in others it could delay the activation of sprinklers needed to suppress the fire. Predicting the response of sprinkler systems in different fire scenarios would provide insight into problem situations. The description of the governing hydrodynamic equations put forth in[1, 2] and the numerical codes in[3] have been expanded to include the effect of sprinkler spray on the fire-induced flow field in an effort to develop a predictive model for sprinkler system response.

The sprinkler/hydrodynamic model is similar to other models such as those described in [4, 5, 6, 7]. For this project, however, the intent is not necessarily to simulate in detail the two phase interaction of droplets and air from a single sprinkler, nor to predict the suppression of the fire itself, but rather to study the effect of dozens of sprinklers on a fire-driven flow field in enclosures up to 60 meters on a side and 10 meters high. The sprinkler spray serves to cool the upper layer hot gases by mechanical mixing with cooler gases below and absorption of heat by the droplets. The approximations to the governing equations described in[1, 2] make such calculations possible on current generation computer workstations.

2 Hydrodynamic Model

Our approach to field modeling fire phenomena emphasizes high spatial resolution and efficient flow solving techniques. This approach has been fostered by the ever-increasing power of computers and the development of faster numerical algorithms. To make the most of the current generation workstations, we have focused our efforts on developing relatively simple numerical algorithms to address the transport of combustion products in relatively simple enclosures. This includes multi-room enclosures, gridded uniformly in the horizontal directions, with the option of variable gridding in the vertical direction. Great efficiency is obtained by embedding the enclosures within either rectangular or cylindrical domains, creating walls and doorways by masking off grid cells. Because simulations involving over a million grid cells are not difficult, fairly elaborate geometries can be considered without sacrificing the spatial resolution, which is two orders of magnitude less than the characteristic length of the enclosure.

The mixing and transport of combustion products is calculated directly from an approximate form of the Navier-Stokes equations. This approximation involves the filtering out of acoustic waves while allowing for large variations in temperature and density [1]. This gives the equations a highly elliptic character, consistent with low speed, thermal convective processes. In fact, a Poisson equation is solved at each time step in the calculation with an FFT-based direct solver [3]. This represents a fair amount of computational work, but the alternative is even less appealing. Solving the full set of Navier-Stokes equations requires the tracking of information at speeds an order of magnitude greater than those usually associated with fire. By filtering out acoustic waves (*i.e.* assuming the sound speed to be infinite), the time step size is limited only by the spatial resolution of the underlying grid. General-purpose solvers often yield unphysical results due to the presence of pressure waves reflecting off of open boundaries. By exploiting the elliptic nature of the flow equations, it is possible to avoid this type of problem.

The fire itself is prescribed in a manner consistent with a mixture fraction based approach to combustion, but the combustion phenomena themselves are not simulated. No turbulence models are employed; the large scale eddies are simulated directly and sub-grid scale motions are suppressed. This is what is meant by the use of the phrase "large eddy simulation". The transport of smoke is simulated by tracking a large number of Lagrangian elements, which originate in the fire. These same elements carry the heat released by the fire, providing a self consistent description of the smoke transport at all resolvable length and time scales. Large temperature and pressure variations are permitted, subject to the limitation that the Mach Number is less than one. The methodology has been used to study a number of fire-related phenomena, for example, the interaction of draft curtains and sprinkler sprays, flows through vents, and more basic combustion problems.

Presently, these computations require approximately 20 microseconds per cell per time step on an IBM RS 6000/58H server, and use several hundred megabytes of memory. The large memory requirement is not a problem, since memory is now relatively inexpensive. With this speed, computations involving over a million grid cells can be done in about 24 hours. Simulations with only tens of thousands of grid cells (usually the limit of most commercial packages) can be done in less than one hour.

The exchange of momentum and energy between a sprinkler spray and a fire-induced flow is computed using a standard model for spherical drag such as that found in Ref. [8] and heat transfer presented in Ref. [9]. Corrections provided by enhanced models for drag and heat transfer [10, 11] were found to be small relative to the uncertainty in the initial water droplet velocity and size distributions.

The two forces influencing droplet motion are gravity and drag. The droplet motion equation is

$$\frac{d}{dt}(m_d \mathbf{v_d}) = m_d \mathbf{g} - \frac{1}{2}\rho c_d A_d \mathbf{v_d}|\mathbf{v_d}| \tag{1}$$

where c_d is the drag coefficient, A_d, $\mathbf{v_d}$ and m_d are the droplet's cross-sectional area, velocity and mass, respectively, and ρ is the gas density. The flow field generated by an activated sprinkler is computed before a simulation begins by computing trajectories for several thousands of droplets. Each trajectory is representative of many more water droplets depending on the sprinkler flow rate. This flow field is then used to define the external force term in the momentum equation. It is assumed that the fire-induced flow field does not significantly influence the trajectories of the droplets,

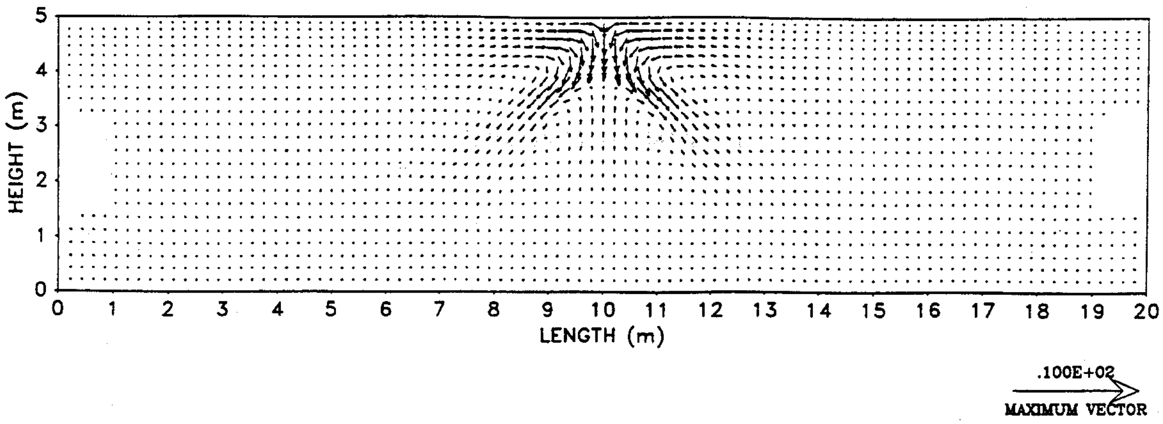

Figure 1: Slice through a simple three dimensional simulation of a single sprinkler, flush with the ceiling in the middle of the open enclosure. Shown are velocity vectors of the gas being entrained into the sprinkler flow. The velocity of the fluid being entrained is on the order of 1 m/s.

whereas the water droplets certainly effect the flow field.

Sprinklers are activated when their link temperature exceeds a certain threshold. The temperature of the sensing element found in heat detectors and sprinklers is estimated using the differential equation developed in [12]

$$\begin{aligned} \frac{dT_L}{dt} &= \frac{\sqrt{|\mathbf{u}(t)|}}{\mathrm{RTI}}\left(T_g(t) - T_L(t)\right), \\ T_L(0) &= T_g(0) \end{aligned} \tag{2}$$

where T_L, T_g are the link and gas temperatures in °C , $|\mathbf{u}|$ is the magnitude of the gas velocity and RTI $(\mathrm{ms})^{0.5}$ is a measure of the sensor's sensitivity to temperature change (a thermal inertia). This model assumes that forced convection is the dominant mode of heat transfer. Heat loss due to radiation and conduction is assumed to be small.

3 Preliminary Computational Results

Shown in Figs. 1 and 2 are two simple examples of the methodology. The first displays the flow field generated by a single sprinkler with no fire-driven flow present. Note the entrainment of air into the sprinkler spray, a key mechanism by which hot gases in the upper layer are cooled. This is shown next in Fig. 2, where a plume of hot gases form a ceiling jet which is redirected downwards due to the force of the water spray from an open sprinkler head. The full-scale simulations presently being performed involve enclosures up to 60 m on a side and 10 m high, gridded by more than one million cells. In most cases considered, about 100 sprinklers are included, spaced 3 to 4 m apart.

Shown in Figure 3 is an example of the methodology applied to a portion of a typical commercial warehouse with rack storage and draft curtains. The ceiling height is 7.5 m (25 ft). The draft curtains are 1.8 m (6 ft) deep and divide the simulation region into four quadrants. The fire size is 5.0 MW.

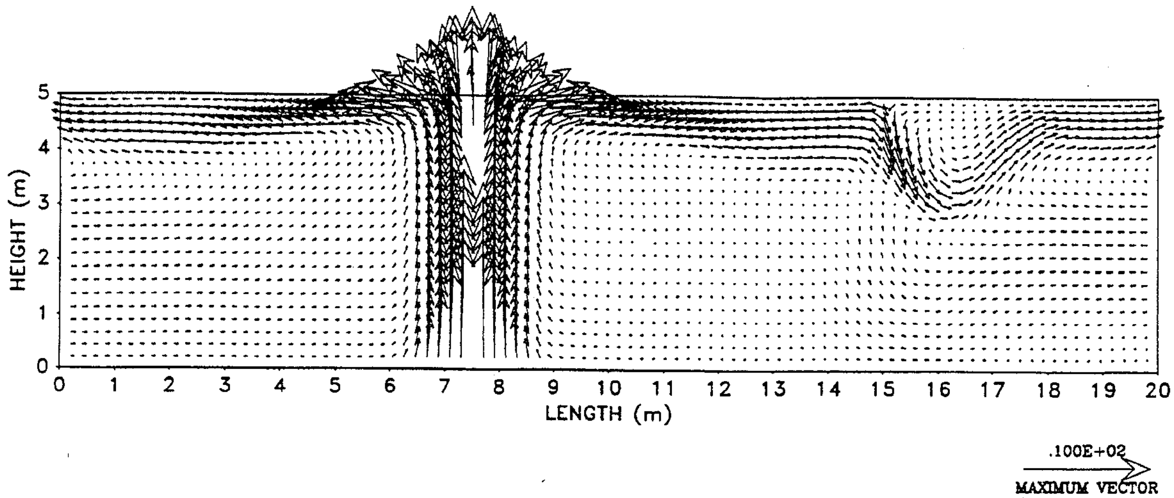

Figure 2: Slice through a simple three dimensional simulation of a ceiling jet impinging on an activated sprinkler (located 15 m from the origin of coordinates).

The fire is located in the upper right quadrant below the center storage rack just to the right of the draft curtain between the two squares labeled 1 and 2.

The sprinklers are modeled using an RTI value of 165 $(ms)^{.5}$ and an activation temperature of 374 K (165 F). The sprinkler links are located 0.18 m (7 in) below the ceiling. The flow rate of each sprinkler was $3.0 \times 10^{-3} m^3/s$ (48 gpm). The order of sprinkler activation is indicated by numbered squares.

Note that all sprinklers in the upper right quadrant (containing the fire) activated. The draft curtains prevented and/or delayed hot fire gasses from going to adjacent quadrants. Only two sprinkler activated in the upper left qudrant. These sprinklers though closer to the fire than most sprinklers in the fire quadrant activated later than all of these sprinklers.

4 Summary

A model for the exchange of momentum and energy between water droplets in a sprinkler spray and gas flow induced by a fire has been incorporated into a field model. A model for sprinkler activation has also been incorporated. Sample cases have been run which illustrate features such as entrainment, gas cooling, that one would expect in a sprinkler/fire flow system.

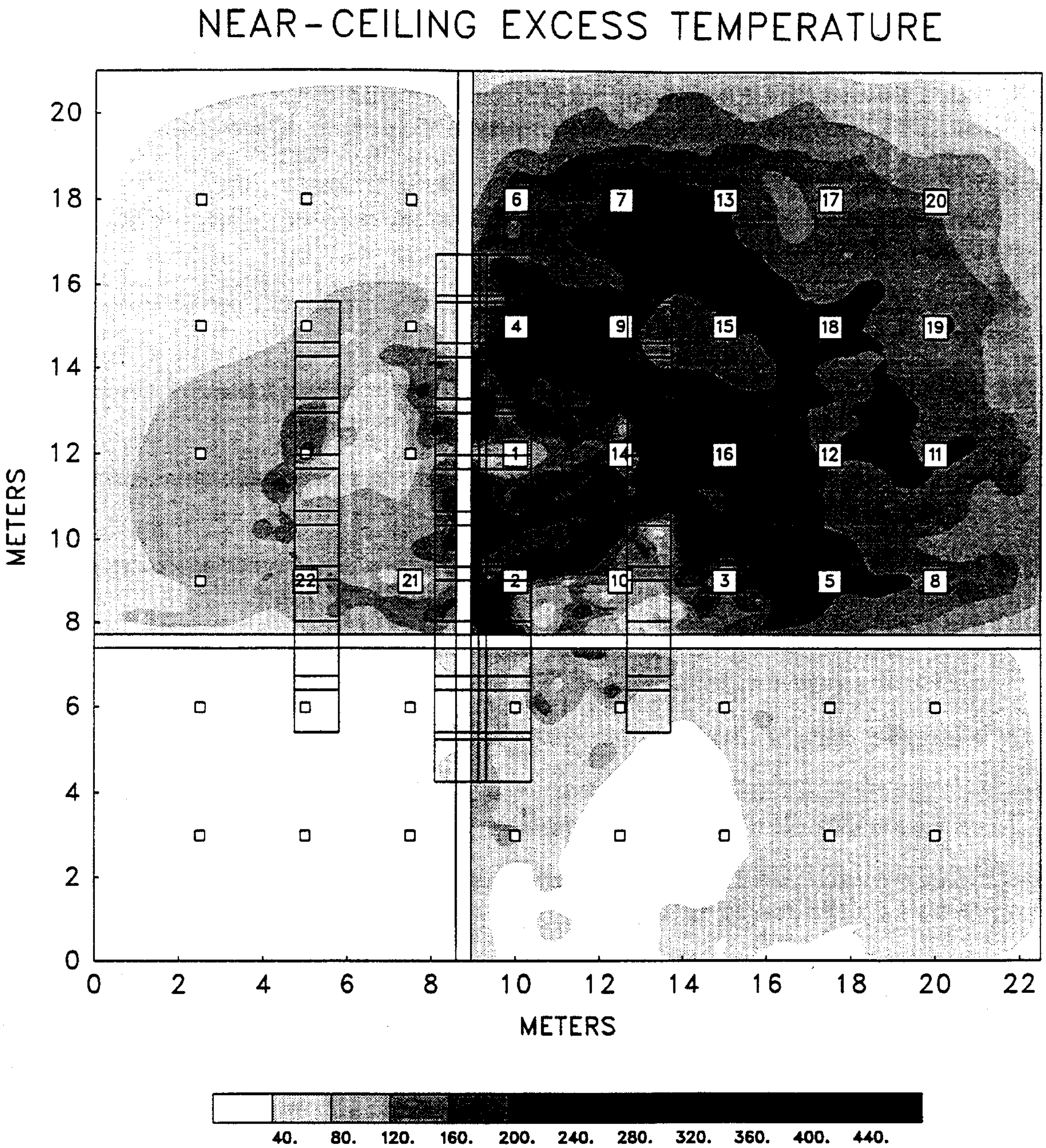

Figure 3: Temperature contour plot in a horizontal slice 0.30 m (1 ft) from the ceiling. The ceiling height is 7.6 m (25 ft). The storage racks are 5.8 m (19 ft) high. The draft curtains are 1.8 m (6 ft) deep. The shaded regions represent temperatures in excess of ambient 100 seconds after ignition. The fire size was 5.0 MW. The numbered squares represent sprinklers in the order that they activated. Squares without numbers represent sprinklers that did not activate. Shown in the figure are three storage racks The fire is located beneath the center storage rack just to the right of the draft curtain. Notice how the draft curtains trap the heat from the other three quadrants.

References

[1] Ronald G. Rehm and Howard R. Baum. The equations of motion for thermally driven buoyant flows. *Journal of Research of the National Bureau of Standards*, 83:297–308, 1978.

[2] H. R. Baum, O. A. Ezekoye, K. B. McGrattan, and R. G. Rehm. Mathematical modeling and computer simulation of fire phenomena. *Theoretical and Computational Fluid Dynamics*, 6:125–139, 1994.

[3] Kevin McGrattan, Ronald Rehm, and Howard Baum. Fire-driven flows in enclosures. *J. Comp. Phys.*, 110(2):285–291, 1994.

[4] Howard P. Morgan. Heat transfer from a buoyant smoke layer beneath a ceiling to a sprinkler spray, 1 - a tentative theory. *Fire and Materials*, 3:27–33, 1979.

[5] R. L. Alpert. Numerical modeling of the interaction between automatic sprinkler sprays and fire plumes. *Fire Safety Journal*, 9:157–163, 1985.

[6] Joseph M. Prahl and Bruce Wendt. Discharge distribution performance for an axisymmetric model of a fire sprinkler head. *Fire Safety Journal*, 14:101–111, 1988.

[7] W. K. Chow and N. K. Fong. Numerical simulation on cooling of the fire-induced air flow by spinkler water sprays. *Fire Safety Journal*, 17:263–290, 1991.

[8] Hermann Schlichting. *Boundary-Layer Theory*. McGraw-Hill Inc., New York, 1979.

[9] Frank P. Incropera and David P. De Witt. *Fundamentals of Heat and Mass Transfer*. John Wiley and Sons, New York, third edition, 1990.

[10] M. C. Yuen and L. W. Chen. On drag of evaporating liquid droplets. *Combustion Science and Technology*, 14:147–154, 1976.

[11] M. C. Yuen and L. W. Chen. Heat-transfer measurements of evaporating liquid drops. *Int. J. Heat Mass Transfer*, 21:537–542, 1978.

[12] Gunnar Heskestad and Herbert F. Smith. Investigation of a new sprinkler sensitivity approval test: The plunge test. Technical Report Serial No. 22485 2937, Factory Mutual Research Corporation, Norwood, MA, 1976. RC 76-T-50.