ABSTRACTS: ANNUAL CONFERENCE ON FIRE RESEARCH

October 13-15, 1992
Rockville, MD

Sheilda B. Smith, Editor

Building and Fire Research Laboratory
Gaithersburg, Maryland 20899
ABSTRACT: ANNUAL CONFERENCE ON FIRE RESEARCH

October 13-15, 1992
Rockville, MD

Sheilda B. Smith, Editor
TABLE OF CONTENTS

INTRODUCTION ............................................................ 1

SESSION 1  Flame Spread


A Model of Low-Speed Concurrent-Flow Flame Spread Over a Thin Solid Fuel, Paul Ferkul and James Tien, Case Western Reserve University ................. 7

Development of an Integral Model for Prediction of Flame Spread on Horizontal Shipboard Surfaces, Vahid Motavalli, Yonggang Chen, Worcester Polytechnic Institute and Michael Delichatsios, Factory Mutual Research Corporation........................................... 9

Flame Spread in Microgravity - Part I: Experimental Study of Simultaneous Upstream and Downstream Propagation, S.L. Olson, NASA Lewis Research Center ................................................................. 11

Flame Spread in Microgravity - Part 2: Theoretical Study on Transition from Ignition to Flame Propagation, Kazuyoshi Nakabe, Howard Baum, Takashi Kashiwagi, BFRL/NIST ........................................... 13

Experiments on Turbulent Upward Flame Spread Under External Radiation, Ellen Brehob, Johnny Bao and Anil Kulkarni, Pennsylvania State University ................................................................. 15

Turbulent, Concurrent, Ceiling and Floor Flame Spread: The Effects of Buoyancy and Oxygen Concentration, L. Zhou, Y.H. Chao and A.C. Fernandez-Pello, University of California ................................................................. 17

Transient Laminar Wind-Aided Flame Spread Over Vaporizing Solids in the Ceiling Configuration, Sanjay Agrawal and Arvind Atreya, Michigan State University ................................................................. 19

Calculating Flame Spread on Horizontal and Vertical Surfaces, Gamal Ahmed, Portland Cement Assoc.; Mark Dietenberger, Forest Product Laboratory; Walter Jones, BFRL/NIST .............................. 21
Critical Conditions for Flame Spread and a Rational Fire Hazard Material Classification Method, Michael Delichatsios, Y. Chen and Mary Delichatsios, Factory Mutual Research Corporation .............................................. 23

SESSION II-A Fire Signatures

Fire Resistant Polymer Crafts, Marc Nyden, James Brown, BFRL/NIST and Bany Bauer, NIST .................................................. 25

Acoustic Emission of Structural Materials Exposed to Open Flames, William Grosshandler and Margaret Jackson, BFRL/NIST ................................. 27

Machine Vision Fire Detection, Douglas Schwartz, Tyndall Air Force Base ................. 29


Session II-B Compartment Flows

Flow Through Horizontal Vents In Enclosure Fins, Y. Jaluria and Q. Tan, Rutgers University .................................................................................. 33

Experimental Study of the Exchange Flow Through a Horizontal Ceiling Vent in Atrium Fins, Tokiyoshi Yamada, Fire Research Institute (Japan) and Leonard Cooper, BFRL/NIST ........................................................................ 35

A Model of Combustion and Convection in Enclosures of Complex Geometry, Kevin McGrattan, Ronald Rehm and Howard Baum, BFRL/NIST ........................................................................ 37

Modeling Backdraft Phenomena, C.M. Fleischmann, P.J. Pagni and R.B. Williamson, University of California ............................................................... 39

What We Know and Don't Know About Ceiling Jets, Howard Emmons, Harvard University ..................................................................................... 41

Local Effects of Vents, Beams and Sprinkler Sprays on the Ceiling Jet Flow, Haukur Ingason, Swedish National Testing and Research Institute .......... 43

SESSION III Flame Structure

Entrainment in the Flaming Region of Fire Plumes, E.E. Zukoski, California Institute of Technology ................................................................. 45


Momentum-Diffusion Controlled Laminar Diffusion Flames, H. Ban, S. Venkatesh and K. Saito, University of Kentucky ........................................... 49

Multiplicity of Puffing Frequencies in Buoyant Diffusion Flames, Baki Cetegen and Tarek Ahmed, University of Connecticut ........................................ 53

A Thermal Element Model for Combustion Processes in Fires, Ofodike Ezekoye and Howard Baum, BFRL/NIST .................................................. 55

Comparison of Experimental and Computed Species Concentration and Temperature Profiles in Laminar, Two-Dimensional CH₄/Air Diffusion Flames, Kermit Smyth, Thomas Norton, BFRL/NIST; J. Houston Miller, George Washington University; and Mitchell Smooke, Yale University ............................................. 57

Numerical Simulation of Thermal Plumes, Soonil Nam and Robert Bill, Jr.; Factory Mutual Research Corporation ........................................ 59

The Turbulence Structure of Medium-Scale Pool Fires, E.J. Weckman and A.B. Strong, University of Waterloo .................................................. 61

Radiative Properties of an Unsteady Jet Diffusion Flame, Carolyn Kaplan, Seung Baek and Elaine Oran, Naval Research Laboratory .................. 63

SESSION IV-A Soot and CO Formation

Additive Effects on Soot Formation in Laboratory Pool Fires, E.J. Weckman and C. Salvador, University of Waterloo .......................... 65

Modeling the Growth of PolynuclearAromatic Hydrocarbons in Diffusion Flames, J.H. Miller, George Washington University ........................................ 67


Carbon Monoxide and Soot Particle Formation Studies in Underventilated Laminar-Diffusion Flames, S.G. Leonard, R. Puri, R.J. Santoro, Pennsylvania State University; and G.W. Mulholland, BFRL/NIST ........................................ 71

A Model of Soot with Detailed Chemical Kinetics, C. Yam and Ian Kennedy, University of California, Davis ........................................ 73

The oxidation of Carbon Monoxide and Soot in Hydrocarbon Diffusion Flames, Rahul Puri, Marlow Moser, Robert Santoro, Pennsylvania State University; and Kermit Smyth, BFRL/NIST ........................................ 75
Measurements of Carbon Monoxide Concentration and Temperature in Axially-Symmetric Methane and Ethylene Diffusion Flames, J.H. Miller, Reed Skaggs and Bijan Alvazl, George Washington University. ............................ 77

Carbon Monoxide Production in Compartment Fires, Nelson Bryner, Erik Johnsson, William Pitts, BFRL/NIST ......................................................... 79

Applicability of the Global Equivalence Ratio for predicting Carbon Monoxide Generation in Enclosure Fires, William Pitts, BFRL/NIST .......................... 81

SESSION IV-B Flame Extinction

Halon Replacement Extinguishing Concentration Requirements: Experiment and Model, Ronald Sheinson and Samuel Baldwin, Naval Research Laboratory ................. 83

Flame Suppression Efficiencies of Next Generation Fire Extinguishing Agents, A.P. Hamins, W.L. Grosshandler and R.G. Rehwoldt, BFRL/NIST .......................... 85

Flame Structure of Inhibited Counterflowing Flames, I.K. Puri, University of Illinois at Chicago and A. Hamins, BFRL/NIST ................................................ 87

Concentration Profiles of Extinguishants in a Fire Environment; Gas Sampling and Analysis, H.G. Eaton, W.D. Smith, J. St. Aubin, R. Brown, H. Burchell, G. Salmon and R.S. Sheinson, Naval Research Laboratory .......................... 89

Quantification of Parameters in the Assessment of Halon 1301 Replacement, W.D. Smith, H.G. Eaton, R. Brown, G. Salmon, H. Burchell, J. St. Aubin and R.S. Sheinson, Naval Research Laboratory ........................................... 91

Theoretical Investigation of C-H and C-X Bond Energy Correlations With OH Reactivity and Flame Suppression Efficiency, Marc Nyden, BFRL/NIST and Karl Irikura, CSTL/NIST ......................................................... 93

Discharge Characteristics of Fire Extinguishing Agents, J.C. Yang, G. Linteris, WL Grosshandler, C. Presser and P. Espina, NIST ............................................. 95

Chemical and Physical Effects of Water During Fire Suppression, Arvind Atreya, Todd Crompton and Sanjay Agrawal, Michigan State University ........................................... 97

Multi-Droplet Evaporative Cooling: Experimental Results, H. Dawson and M. di Marzo, University of Maryland .......................................................... 99

SESSION V-A Soot and Radiation

Modeling of Turbulent Flame Radiation, J. de Ris and L. Orloff, Factory Mutual Research Corporation ................................................................. 101

Wall-Fire Radiant Emission, G.H. Markstein and J. de Ris, Factory Mutual Research Corporation ................................................................. 103
Heat Feedback to a Pool Fire, A. Hamins, BFRL/NIST; M. Klassen, J. Gore, Purdue University; and T. Kashiwagi, BFRL/NIST ................................. 105

The Role of Reflection and Fuel Vapor Absorption in Radiative Heat Feedback to Pool Flames, M. Klassen, Y.R. Sivathanu, J.P. Gore, Purdue University .............. 107


Specific Absorption Coefficients of Soot Particles in Laminar Flat Flames, Y.R. Sivathanu and J.P. Gore, Purdue University .............................. 111

Scattering by Soot Aggregates in Turbulent Diffusion Flames, R. Mital, J. Dylla, Y.R. Sivathanu and J.P. Gore, Purdue University .......................... 113


SESSION V-B Pool Fires: Large Fires

Anchoring Mechanism of Pool Fires, S. Venkatesh, A. Ito and K. Saito, University of Kentucky ................................................................. 117

Effect of Pressure and Oxygen Concentration on Pan Fires in an Enclosed Space, Jean Bailey, Patricia Tatem and Frederick Williams, Naval Research Laboratory ..................................................................... 119

Fire-Induced Winds in the 1991 Oakland Hills Conflagration, Javier Belles and Patrick Pagni, University of California ........................................... 121

........................................................................................................ 123

A Lumped Parameter Model for Unburned Tree Crown Streets, M.C. Smith, Michigan State University and Warren Heilman, U.S. Forest Service .... 125

Numerical Simulation of the Rise and Dispersion of Fire Generated Plumes, Ahmed Ghoniem and Xiaoming Zhang, Massachusetts Institute of Technology ........................................................................... 127

Two-Dimensional Simulations of Buoyancy-Induced Horizontal Roll Vortices, Warren Heilman, USDA Forest Service ........................................ 129
**POSTERS**

*Experimental Study of Forced Ventilation Glovebox Fires*, Craig Beyler, Hughes Associates; Michelle Peatross and Richard Roby, Virginia Tech ................. 131

*Corridor Flow of Smoke from Postflashover Fire: a post."FIRE SIMULATOR" Module for FPETOOL*, Harold Nelson and Scot Deal, BFRL/NIST ................. 133

*Flame Retardancy of Cotton • Synthetic Blend Fabrics*, Robert Harper, Jr., George Ruppenicker and A.P.S. Sawhney, USDA Southern Regional Research Center, USA. ........................................ 135

*Direct Measurement of Heat Release Rates*, C. Vwelle, J.L. Delfau, E. Moussan, CNRS and C.Pham Van Cang, EDF, France ........................................ 137

*Hot Surface Ignitions: Ionic Effects in Ignition Initiation*, William Bannister, University of Massachusetts and Douglas Nelson, Tyndall Air Force Base ........................................ 139

*Experimental Study of Self-Ignition of Electrical Cables Exposed to an External Radiant Flux*, Marie-Claude Lopez and Jacques Bergelin, IPSN/CEA, France ........................................ 141

*ARGOS: A Computer Program for Fire Risk Evaluation*, Niels Baden, Danish Institute of Fire Technology ........................................ 143

*Analysis of the National Fire Protection Association Fire Risk Assessment Framework: Example Calculation*, E Merrill Galloway, BF Goodrich Technical Center and Marcelo Hirschler, Safety Engineering Laboratories ........................................ 145

*An Experimental Investigation of the Pulsation Frequency of Flames*, A. Hamins, J.C. Yang and T. Kashiwagi, BFRL/NIST ........................................ 147

*Acquisition and Analysis System of Burn Injuries From Flames and Thermal Radiation*, Barry DeCristofano, Gerald Caldarella, Brian Kimball and Joseph Roach, U.S. Army Natick RD&E ........................................ 149

*Interrelationship Between Hydrogen Cyanide Gas Exposure Concentration, Exposure Time and Blood Cyanide Level in Rats*, Arvind Chaturvedi, Boyd Endecott, Roxane Ritter and Donald Sanders, FAA ........................................ 150

*Performance of the Reciprocal Nephelometer*, George Mulholland and Nelson Bryner, BFRL/NIST ........................................ 152

*Correlations for Temperature and Velocity in Room Fires for Estimating Sprinkler Activation Times*, Pravinray Gandhi, Underwriters Laboratories ........................................ 154
Sprinkler Head Response Measured and Predicted, Haukur Ingason, Swedish National Testing & Research Institute .......................................................... 156

Thermal Radiation From Marine Fire Boundaries, LeMoyne Boyer, Southwest Research Institute. .......................................................... 158


One and Two-sided Burning of Thermally-Thin Materials, T. Ohlemiller and J. Shields, BFRL/NIST ................................................................. 164

A Comparison of Numerical and Exact Solution of the Creeping Flame Spread Over Thermally Thin Material, Subrata Bhattachajee, San Diego State University ................................................................. 166

The Behavior of High Density Cellulose Samples Under Simulated Fire Conditions, I. Milosavljevic and E.M. Suuberg, Brown University .............. 168
INTRODUCTION

The MST Annual Conference on Fire Research has long been the prime forum for the presentation and discussion of the latest advances in the science of fire and the engineering of fire safety. This year's Conference has been expanded to include all fire research performed within Federal laboratories or sponsored by Federal agencies, as well as by laboratories around the world. The Conference was organized with the assistance of representatives from those Federal agencies:

Mr. Donald Bathurst  General Services Administration
Mr. Michael Bennett  Wright-Patterson Air Force Base
Dr. Thor Eklund  Federal Aviation Administration
Mr. Kenneth Faulstich  Department of Veterans Affairs
Dr. Robert Friedman  NASA Lewis Research Center
Ms. Karla Kertis  U.S. Bureau of Mines
Dr. Merrill King  National Science Foundation
Mr. Dennis Kubicki  Department of Energy
Mr. Roger Lanahan  U.S. Fire Administration
Mr. Domenic Macaione  U.S. Army Materials Technology Laboratory
Mr. Charles McGuire  Department of Transportation
Mr. Jonas Morehart  National Institutes of Health
Ms. Margaret Neily  Consumer Product Safety Commission
Mr. Gene Sober  NASA Goddard Space Flight Center
Dr. Patricia Tatem  Naval Research Laboratory
Mr. Richard Vickers  Tyndall Air Force Base
Dr. Robert White  Forest Products Laboratory

This booklet contains the abstracts of the 64 papers and some 24 posters focussing on the phenomenology of fire: flame spread, flame structure, CO and soot formation, soot radiation, fire signatures, flame extinction, compartment flows, and large fires. Discussion sessions will consider the status of our knowledge and the most important understanding yet to be developed. With this, we hope to continue cross-pollinating the elements of the fire research community while stimulating our members to better our knowledge base for reducing the toll from unwanted fires.

Richard G. Gann, Conference Chair
Chief, Fire Measurement and Research Division
Building and Fire Research Laboratory
National Institute of Standards and Technology
ROLE OF IN-DEPTH ABSORPTION IN FLAME SPREAD UNDER EXTERNAL RADIATION OVER SEMITRANSPARENT MATERIALS

by S. S. Manohar, A. K. Kulkarni and S. T. Thynell
The Pennsylvania State University

When radiation is incident on a semitransparent material, it is partially reflected and partially transmitted through the surface, which is then absorbed volumetrically within the material itself. Plastics, such as polymethylmethacrylate (PMMA), are often highly semitransparent. One of the practical problems in which the in-depth radiation absorption can be important is flame spread over semitransparent solid fuels, such as a burning wall or floor with spreading flame in the presence of a surrounding fire. Flame spread and surface heating under external radiation has been studied theoretically, however, the models have used either opaque solid formulation [1] or gray in-depth radiation absorption [2]. When in-depth absorption is ignored, all the energy is assumed to be deposited on the surface. This results in a more rapid rise in surface temperature and consequently in a greater rate of predicted flame spread. The objective of the present work is to study the effect of in-depth radiation absorption on the surface temperature of a semitransparent material, using PMMA as example.

Mathematical Model: The medium is modeled as a one-dimensional slab, insulated at one end and subjected to external radiation at the other. To examine the effect of in-depth radiation absorption the material is then considered to be either semitransparent or opaque. In a real fire situation, the surface well ahead of the pyrolysis front may be clear initially, but later it gets covered with a layer of soot as the combustion progresses. Hence, two cases are considered, one where the surface is clear and the other where the surface is covered with a layer of soot, for each of the semitransparent and opaque formulations. In order to accurately solve the radiation problem, a 14-band model is employed to simulate the spectral distribution of absorption coefficient and reflectivity, which were determined experimentally.

Experiments: Two series of experiments were conducted, radiation property measurements for input to the model, and transient surface temperature measurements for a slab in the presence of strong external radiation to verify the model.

Absorption Coefficient and Reflectivity Measurements: The spectral radiation properties were measured using a standard Fourier Transform infrared (FTIR) spectrometer. A wavelength range from 0.67 μm to 20 μm was covered using two different combinations of detector and beam splitter, a potassium bromide beam-splitter with a mercury cadmium telluride (MCT) detector and a lead selenide detector with a quartz beam splitter. With the measurement of the spectral transmittance, the transmissivity, $\tau_\lambda$, and reflectivity, $\rho_\lambda$, were obtained using available procedure.

Figure 1 shows absorptivity data for clear PMMA samples. It can be seen that the absorption coefficient is low and highly irregular in the range 1000 to 3000 cm$^{-1}$, which may be explained by examining the molecular structure of PMMA. Also shown in Fig. 1 is the 14-band model for the absorption coefficient. The net effect on energy absorption depends on both the spectral distribution of incident radiation and the spectral properties of the absorbing medium. Therefore, a carefully selected multiband radiation model is needed to accurately account for the in-depth absorption. The radiation received
by a burning wall or floor is typically from surrounding fire which contains very little amount of energy below 2000 cm\(^{-1}\) and the property data show absorption coefficient diminishing to almost zero above 6000 cm\(^{-1}\). Therefore, the range between 1775 cm\(^{-1}\) and 6250 cm\(^{-1}\) was divided in 14 bands. The spectral radiation properties of absorptivity and reflectivity property data are then used as an input to the mathematical model.

**Surface Temperature Measurements:** Experiments were carried out to determine the rise of surface temperature on two sets of PMMA samples, those with clear surface and those with soot-layer covered surface. The samples were then subjected to external radiation at five different heat flux levels, ranging from 3.7 to 23.15 kW/m\(^2\). Experimental data were recorded until either the steady state was reached or the surface of the sample was completely covered with bubbles.

**Model results and comparison with experiments:** The predictions were compared with data for the four combinations of semitransparent and opaque material slabs with clear and blackened surfaces at different values of external heat flux. As an example, comparisons for blackened samples are shown in Fig. 2 at 6.8 kW/m\(^2\). The difference between the opaque and semitransparent case represents the effect of in-depth radiation absorption. It is clearly seen that it takes longer for a semitransparent sample to reach a fixed temperature than the opaque sample.

The results have important implications for predicting flame spread under external radiation because the rate of flame spread depends upon the rate at which the surface just ahead of the pyrolysis front reaches a critical temperature. If the material is semitransparent, the model must adequately account for in-depth absorption, otherwise, an accelerated flame spread will be predicted. Additional details on the present work may be found in reference [3].

**References:**

This work was partially funded by a grant from the National Institute of Standards and Technology.

**Figure 1** Absorption coefficient and 14 band model

**Figure 2** Effect of in-depth absorption on surface temperature rise of blackened samples
MATHEMATICAL MODEL OF COCURRENT HORIZONTAL FLAME SPREAD OVER SOLID COMBUSTIBLES

C. C. Hwang and C. D. Litton
Pittsburgh Research Center
U.S. Bureau of Mines
Pittsburgh, PA 15236-0070

Introduction

The flame spread along thermally thick fuels in the presence of a co-current air stream is in general more rapid and more hazardous than the spread in opposed-flow configurations. The interest in fire safety for co-current mode of flame spread motivated the development of flamability test methods [Steiner, 1943; Parker, 1977; Perzak and Lazzara, 1992]. de Ris (1970) obtained analytical solutions to fuel-rich fires spreading along fuel-lined ducts. The temperature distributions in the gas and the walls were obtained for the preheating zone, the combustion zone, and the burn-out zone. In the existing literature on the flow-assisted flame spread, the rate of flame spread is a monotonic function of the air speed [Fernandez-Pello, 1979; Zhou and Fernandez-Pello, 1990]. This is in contrast to our observation with the fire spread along conveyor belts that a maximum spread rate occurs at the air velocity approximately 1.5 m/s. In the experimental setup in a large-scale gallery, the hot gas plume appears to entrain air from all sides, i.e., the flow configuration is three-dimensional. In addition, the combustion gas stratifies and the radiative heat transfer is the dominant mode of energy transfer. This is in contrast to the flame configuration in small-scale fires, i.e., the flow field is approximately two dimensional and the convective heat transfer is the main mode of energy transfer. In these cases, the air is entrained into the flame zone from the rear of the flame. It appears that the traditional, two-dimensional, boundary-layer type approach cannot model the flame spread for the conditions encountered in large-scale gallery fires.

Mathematical Model

The spread of a flame along a horizontal bed of solid fuels is considered in the present study. The direction of the air flow is same as that of the flame spread, and the pyrolyzed vapor from the solid combustibles and the combustion gases are convected ahead of the pyrolysis front enhancing the transfer of heat and thus assisting the spread of the flame. The spread of flame is controlled by radiative heat transfer from the flame zone to the fuel and by convective heat transfer from the combustion gas to the fuel. The heat energy entering the fuel surface is transmitted by conduction. The heating causes the fuel temperature to increase until it reaches the pyrolysis temperature. The pyrolyzed matter releases its chemical energy in the flame zone. The air in the vicinity of the flame zone is entrained into the zone to provide the required oxygen. The amount of entrained air is computed from the equations given by Delichatsios (1988). The air stream forces the flame to tilt toward the unburnt fuel. The tilt angle is computed from the equations given by Mudan and Croce (1988). The flame temperature is calculated from an energy balance for a control volume enclosing the flame zone. The soot volume fraction which is employed in the calculation of the flame emissivity is obtained from an experimental correlation.

At each time step, the surface temperature and the pyrolysis rate are computed for each cell of the fuel. In the present analysis, the flame front is the location on the flame surface where the temperature reaches its pyrolysis temperature. The burn out of a cell occurs when a fraction of the cell mass is consumed. Three flame volume shapes have been considered, a parallelepiped, a prism and a cylinder. The results presented are based on the prism-shaped flame volume.

Results and Discussions

In the modeling, a fire is initiated from a length of the combustible material. For a given air speed, there is a minimum initial length below which the flame does not propagate. The length of the burning zone may or may not reach a final ‘steady length’ during a burn. Figure 1 shows the flame position versus time for several values of the air velocity. In most cases the spread has not reached a stational...
Figure 2 plots the spread rate as a function of the air velocity. Line A is obtained based on the mean spread rate for the total burn time shown in Figure 1. A maximum spread rate is indicated at the air velocity of approximately 1.5 m/s. Line B is calculated from the near-straight portions of lines in Figure 1. For this case a maximum spread rate is at the air velocity of approximately 2.5 m/s. The experimental data are from sheets of 12.5 mm thick, 0.65 m wide by 2.4 m long PMMA and 12.5 mm thick, 1.0 m wide by 10 m long styrene butadiene rubber (SBR) conveyor belt. It may be noted that the lengths of the fuels used in the experiments are short, and station values of spread are unlikely to be reached. In general, the model predicts the overall trend of the flame spread observed in large-scale gallery experiments.

References


Figure 1: Flame front position vs time.

Figure 2: Flame spread rate vs air velocity.
A MODEL OF LOW-SPEED CONCURRENT-FLOW FLAME SPREAD OVER A THIN SOLID FUEL

Paul V. Ferkul and James S. T'ien
Department of Mechanical and Aerospace Engineering
Case Western Reserve University

Abstract

In spacecraft cabins, low-speed flows are generated due to ventilation requirements. The flows can be as low as a few centimeters per second, which is below the velocity range typically encountered in buoyant fires on earth. The flame propagation and the extinction mechanisms in this low-speed regime can be quite different from those on earth. Both the concern of spacecraft fire safety and curiosity about fundamental scientific understanding of flame behavior in low-speed flows prompts this investigation.

The model considers two-dimensional, steady flame spread in concurrent flow over a thin solid. Because of its thinness, the fuel bed can burn through and the flame is capable of reaching a constant length. Thus, following the flame, a steady state, with constant (and equal) flame tip velocity, pyrolysis front, and fuel burnout rates, can be obtained.

The flame is assumed to be laminar because of its small size and low velocity. The gas-phase fluid mechanical treatment is more comprehensive than most works in the past. In the flame base region (where the upstream flow is seen first), streamwise heat and mass diffusion is included. This, together with finite-rate gas-phase chemical kinetics, enables us to examine the question of flame stabilization and extinction. The elliptic treatment of this flame stabilization zone entails the full Navier-Stokes equations. On the other hand, to save computational time, a boundary layer approximation is employed in the downstream region. The two zones are coupled at an appropriate gas-phase location, they are also coupled indirectly through the solid by energy exchange. A simplified solid fuel model is used. The solid is assumed to be thermally thin and pyrolyzes according to a one-step Arrhenius law with no char and tar formation. A solid surface radiative loss term is included which turns out to be of critical importance in the low speed limit. Because of the coupling between elliptic and parabolic regions and between the gas and solid phases, the numerical solutions require many iterations and are computationally intensive; they are carried out using Cray X-MP Supercomputer at NASA Lewis Research Center.

Extensive computations have been performed using oxygen mole fraction and free-stream velocity as parameters. Fig. 1 gives all the points calculated and the extinction boundary. Some of the important results are summarized below.

1. A low velocity extinction limit exists. This is the consequence of surface radiative loss. While the maximum flame temperature remains constant at low velocity without radiative loss, with radiative loss the flame temperature drops with decreasing velocity. Without radiative loss, the model predicts no low-speed extinction limit.

2. Flames quench at low speed by shrinking in size. Near-limit flames remain with the fuel burnout region (flame stabilization zone). This is in contrast to flame blowoff at high speed ($U_\infty > 5$ cm/sec in Fig. 1) where the flame is unable to stabilize in the fuel burnout region and extinction is reached when it is blown downstream.
(3) The minimum oxygen percentage that will support a spreading flame in concurrent flow occurs at $U_\infty = 5 \text{ cm/sec}$ (see Fig. 1) which is well below, the buoyant flow velocity at normal gravity. This can have profound implications for spacecraft fire safety.

(4) The computed reaction rate contours suggest short and fat flames in low speed flow. This is verified by recent drop tower experiments by Grayson. The flame spread rates are generally small, ranging, for example, from 0.08 cm/s to 1.56 cm/s for convective velocities 0.9 to 10 cm/s in twenty-one percent oxygen. These computed spread rates are also roughly close to the experimental results. A more precise comparison cannot be made at the present time due to the time limitations of the drop tower which prevents the achievement of the final steady flames.

(5) Using finite rate kinetics enables us to examine the percentage of fuel which escapes from the flame. As velocity is decreased toward the quench limit, this percentage increases due to increasing degree of flame tip quenching. The presence of unburnt fuel vapor in spacecraft can be a safety concern.

Detailed flame structures for all the cases shown in Fig. 1 have been obtained but cannot be shown here due to the page limitation. Some of the most interesting ones will be presented at the conference.

Acknowledgement

This research is supported by NASA Microgravity Science and Applications Division through Grant NAG 3-1046.

Fig. 1. Extinction Boundary for Concurrent-Flow Flame Spread Over a Thin Solid Fuel
Abstract submitted to: ANNUAL CONFERENCE ON FIRE RESEARCH

Development of an Integral Model for Prediction of Flame Spread on Horizontal Shipboard Surfaces

Vahid Motevalli and Yonggang Chen
Worcester Polytechnic Institute

Michael A. Delichatsios
Factory Mutual Research Corp.

A model for prediction of flame spread on horizontal surfaces is being developed for the Naval Research Laboratory. It is a general model for any horizontal surface, both charring and non-charring material, with a flexibility to incorporate the possible restrictions imposed within a shipboard compartment such as vitiated environment or heat transfer from adjoining compartments. This paper describes the components of the model being developed and the progress made during the first year of this effort.

The components of the Shipboard Horizontal Flame Spread Model (SHFSM) are as follows:

1) An integral transient pyrolysis model for charring and non-charring material,
2) An integral combustion model, and
3) The model skeleton which includes calculation of the heat fluxes arriving at the surface ahead of the flame front.

The transient pyrolysis model is based on a one-dimensional heat conduction equation solved for any arbitrary heat flux at the surface. The model has been tested with very good results for thermally thick-to-intermediate material. In this model, the material is divided into two layers of char and non-pyrolyzed segments and the char front propagation is tracked. The reradiation loss at the surface is accounted directly while the convective heat transfer at the surface is treated through use of equivalent properties. The integral approach and use of robust numerical techniques allows rapid computation. Furthermore, an approach has been developed and refined to use experimental results of a mass loss and surface temperature histories to derive char properties such as thermal conductivity and heat of pyrolysis. This transient model has been incorporated into the preliminary flame spread model.

As the flame spreads on the surface, the fire grows due to increased burning surface. This in turn will affect the rate of flame spread and has to be coupled with the flame spread model. The total heat flux from the flame ahead of itself is directly related to the flame geometry and heat release rate. A combustion model, coupled with the transient pyrolysis model is needed to address this phenomenon.

An integral combustion model has been developed for turbulent jet diffusion flame that includes, as its most important component, the flame radiation and its relation to flame turbulence and the sooting tendency of the fuel. The soot generated by the fuel can be characterized by the laminar smoke-point height. This model relies both on previous integral models and more detailed k-e-g models. The main characteristics of the present integral model are the following:

1) Instead of an entrainment equation, a dimensionless correlation for the entrainment spanning the range from momentum to buoyant turbulent jet flames has been employed and validated.
2) A direct relationship for the fluctuation of a conserved scaler inspired by recent work of Chatwin and Sullivan and validated by recent experiments is also used.
3) In the present model, radial profiles for the mixture fraction and velocity are derived in order to preserve the strong non-linearity of the combustion process.
4) Most importantly, a flame radiation model applicable for luminous (soot radiation) optically thin flames has been developed whose sooting tendency is characterized by the laminar smoke-point height.

In order to characterize the turbulent mixing, a beta (beta) probability for the conserved scaler has been used which has been shown to represent the turbulent mixing process well. The flamelet model, which is based on the approximation of the turbulent diffusion flame as an ensemble of luminous flamelets along with known state relationships (i.e. species concentration as function of mixture fraction)
for the various products of combustion has been applied to develop the combustion model\(^8,9\). These state relationships (some of which are available in chemical libraries) can be obtained by experiments in laminar flames\(^8\) or by detailed chemical kinetics calculations. Radiation is treated here in the optically thin limit (as a first attempt) by making the following approximations:

1) All radiation originates from deviations that are near the instantaneous maximum flame temperature.
2) The effective temperature decreases due to radiant losses as the radiation fraction increases.

The combustion model will provide the flame temperature, radiative loss fraction and major species concentration as a function of height above the fuel surface. These are essential parameters to not only determine the heat flux ahead of the flame, but also provide coupling with the environment as well as input for a full compartment fire model. The combustion model is under development and some promising preliminary results have been obtained thus far.

At the present time, the flame spread model uses the surface temperature reaching the pyrolysis temperature as a spread criterion. The heat flux from the flame and any external flux plus heat losses are used to arrive at a net flux at the surface. The net flux is used as the time-varying (implicitly since the pyrolysis front position, \(x_p = x_p(t)\)) input to the transient pyrolysis model to determine the surface temperature as a function of time. A variable mesh spacing has been shown to produce robust and accurate results. Figure 1 shows an example of the results from the SHFSM for an arbitrary heat flux.

Some preliminary experiments have been designed and performed using the LIFT apparatus in a horizontal orientation to attempt to define the heat flux ahead of the flame.

References
Flame Spread in Microgravity
Part 1: Experimental Study of Simultaneous Upstream and Downstream Propagation
S.L. Olson
NASA Lewis Research Center

Introduction
This summarizes the progress of the experimental portion of a collaborative experimental/numerical effort between NASA and NIST to study flame spread over a thermally-thin cellulosic material in low gravity. Although presently there are differences in conditions between experiment and model such as ignition mode and geometry, both aim to understand the effects of oxygen concentration and low velocity forced flow on flame spread.

Experiment Description
A preliminary series of tests was conducted in the 22 second Drop Tower at NASA Lewis Research Center to study the effects of low velocity flow on flame spread over thermally thin solids. As shown in Figure 1, to simultaneously obtain both upstream and downstream flame propagation in a single test, the 5 cm wide by 15 cm long thermally-thin cellulosic fuel was centrally ignited by a straight nichrome wire run across the full width 75 cm from each end. This planar configuration is similar to the axisymmetric case modelled by Kushida et al [11]; the downstream gas-phase flow and atmosphere is affected by the presence of the upstream flame, which perhaps is more representative of a typical fire scenario than ignition at either end of the sample. Flow over the sample was 0', 2', or 5 cm/s; molar concentrations of oxygen were 21 or 30% in N2.

It should be noted that during the short test times, the flame never fully separated into two independent flames; the flame base of the downstream flame was actually still the upstream flame leading edge. The downstream flame spread rate is defined here as being the movement of the downstream flame tip.

Preliminary Results
Figure 2 shows flame spread rates and position versus time data for the 30% oxygen tests at 0 and 2 cm/s. When compared with previous work on opposed-flow flame spread [2], the upstream flame spread rates measured in these tests are in good agreement; they show the same magnitude and indepen-
The position versus time data and flame spread rates for the 21% oxygen tests at 0 and 5 cm/s are shown in Figure 3. They demonstrate the expected dependence on opposed flow velocity, but the spread rates are slightly faster than previous work [2]. Downstream flame spread rates were not steady in air, and the flame tips appeared to be retreating slightly just prior to the end of the 22 second test.

Both quiescent tests (in air and at 30% oxygen) revealed that the flame spread rates away from the central ignitor are not equal as they should be in a truly zero gravity environment. These results are discussed below.

**Discussion**

The observation that all downstream flames are slower than the upstream flame is the opposite of normal gravity flame spread where co-current flame spread rates are typically faster than opposed-flow flame spread rates. This may indicate that the downstream flame is weakened by the presence of the upstream flame. The retreating flame tips of the downstream flame in the air tests may be an indication that, given longer test times, the downstream flame is not viable under these atmospheric and flow conditions. It is interesting that the downstream flame at 30% oxygen with a 2 cm/s flow is “weaker” than the comparable downstream flame under quiescent conditions. This is the reverse of the upstream flame where an increase in flow velocity in low gravity assists the flame spread process. This may be due to the close matching of the magnitudes of the downstream flame spread rate and flow velocities, so the relative velocity between the two is small, resulting in a reduced convective contribution to heat and mass transport in the flame.

The discrepancy between upstream flame spread rates in air and previous opposed flow flame spread rates [2] may be due to the different boundary layer thicknesses at the 7.5 cm vs 15 cm downstream of the leading edge for the previous work [2]. Boundary layer effects on flame spread have been observed in normal gravity [3]. The non-symmetry in flame spread in the quiescent tests is currently being investigated; the non-symmetry is attributed to a test condition that is not truly quiescent, and two possible sources of flow are 1) the chamber light induced buoyant flow in the chamber, or 2) there is a residual gravity level in the 2.2 second drop tower.

**Planned Work**

The gravity level during a low gravity test in the 22 second drop tower will be measured in the near future. Quiescent tests will be rerun without the light in the chamber. Further flame spread tests are to be conducted in the 5.2 second NASA Lewis Zero Gravity Facility, where longer test times may yield complete separation of the two flame halves or possible extinction of the downstream flame. Boundary layer effects on opposed-flow flame spread will also be investigated further.

**References**

FLAME SPREAD IN MICROGRAVITY : Part 2
Theoretical Study on Transition' from Ignition to Flame propagation

Kazuyoshi Nakabe, Howard R. Baum and Takashi Kashiwagi
Building and Fire Research Laboratory, NIST

Introduction

This presentation describes the progress of the theoretical portion of a collaborative numerical/experimental effort between NASA and NIST to study flame spread mechanisms over a thermally-thin cellulosic material in microgravity. Although at present some differences in conditions such as ignition mode and geometry exist between the studies, both aim to understand the effects of oxygen concentration and forced slow flow velocity on flame spread.

Theoretical Model

An axisymmetric time-dependent model describing ignition and subsequent transition to flame spread over a thermally thin cellulosic sheet in a quiescent zerogravity environment has been developed. This model is currently being extended to a three-dimensional model including a slow external flow along the sample surface. The absence of gravity removes the buoyancy induced vorticity generation mechanism. The small scale of the physical size (characteristic length scale is about 1 cm which is the Gaussian half-width of an external radiation beam) with a quiescent condition or with a slow external flow implies a low Reynolds number flow domain. Due to the unique combination of a zerogravity environment and low Reynolds number flow domain, the model consists of a potential flow calculation coupled to mass, energy, and species conservation equations[1]. A one-step global gas phase oxidation reaction and three global solid phase degradation reactions[2], endothermic pyrolysis, exothermic oxidative degradation and highly exothermic char oxidation reaction, are used in the model.

In the calculation the sample is irradiated and auto-ignited at the center by thermal radiant flux with a Gaussian distribution instead of pilot ignition for the experiment. A maximum external radiant flux of 5 W/cm² at 21%, 30% and 50% oxygen concentrations in a quiescent environment is used in the calculation. At present development of a three-dimensional time-dependent model including an external slow flow is in progress. Some of preliminary results might be presented in the meeting.

The results indicate that ignition is observed for 21% and 30% oxygen concentrations but the transition to flame spread does not occur. However, in 50% oxygen the transition is achieved as shown in Fig.1 which plots gas phase temperature distribution 1.0s after initiation of the external radiation. Each temperature isotherm line is a 100°K interval starting from 350°K. Distributions of velocity vector and oxygen/fuel concentrations at the same time are shown in Figs.2 and 3, respectively. In Fig.3 solid lines are fuel concentrations and dashed lines are oxygen concentrations with an interval of 2%. The energy balance at the sample surface at 1.0s is shown in Fig.4, where $Q_{ex}$ is the external radiant flux, $Q_{rad}$ is the re-radiation loss, $Q_{fcon}$ is the net heat balance of the three degradation reactions, $Q_{mct}$ is the convecting heat transfer from the gas phase to the sample and $Q_{vct}$ is the energy expressed by mass flux times enthalpy difference. A high energy feedback rate of nearly 6 W/cm² is calculated instead of nearly 1 W/cm² for 21% oxygen concentration case at 1.6s after initiation of the external radiation as shown in Fig.5. This
difference in energy feedback rate is mainly caused by differences in flame temperature (about 1400°K for 21% vs 2000°K for 50%) and also in the location of peak gas phase reaction rate (closer to the sample surface in 50% oxygen concentration). Detailed discussion on the transition mechanisms and also comparison with experimental data will be presented in the meeting.

Reference
EXPERIMENTS ON TURBULENT UPWARD FLAME SPREAD
UNDER EXTERNAL RADIATION

by Ellen G. Brehob, Johnny Bao, and Anil K. Kulkarni

The Pennsylvania State University

The overall objective of the present project is to understand the upward flame spread phenomenon under simulated surrounding fire conditions. This is achieved by conducting experiments on upward flame spread under external radiation, developing a mathematical model, measuring the relevant basic material properties needed, and checking the validity of the model by comparing predictions with data. Emphasis is placed on studying and predicting the behavior of practical wall materials used in building and vehicle interior and textiles. This paper presents results of a comprehensive series of flame spread experiments conducted on Particle Board samples at various external radiant heat flux levels, under preheat and no preheat conditions, and for a range of line-burner igniter strengths. In each test, measurements were made as a function of time for total heat feedback and surface temperature at five different heights, and the flame height was read from the video record. Figure 1 shows a schematic of the experiment setup in which wall samples 0.3m wide by 1.2 m high (1'x 4') were subjected to external radiation fluxes of up to 29 kW/m² using two large electrically heated panels.

Effect of External Radiation: Figure 2 shows an increase in flame height as a function of time at six different heat fluxes. The samples were not preheated and the igniter was set at 18 kW/m² in all tests. The external flux not only enhances the upward flame spread significantly, but also affects whether the upward flame spread will be sustained or not. At 5.4 kW/m² and above the flame spread was self-sustained; at heat flux levels tested below that threshold it did not.

Effect of Preheat: When the particle board sample was preheated to steady state and then ignited, the flame spread was substantially greater compared to the sample which was simultaneously ignited and subjected external heat flux. (See Fig. 3.) Here, the heat flux was 5.4 kW/m², and the front and back surfaces had reached an average temperature of 142°C and 76°C, respectively. Also shown in the figure is the scatter in the measurement of the flame height data, which was present in all the tests.

Effect of Igniter Strength: The igniter strength determines the height of the flames from the line burner. A large ignition source can produce sufficient size of the initial pyrolysis zone to sustain the upward spread. Figure 4 shows interesting results of the effect of the igniter strength. The largest igniter flames clearly produced a sustainable fire, the smallest failed to do so, and the middle size initially created flames up to the top, however, the flames became intermittent and then dropped back to a much lower level.

Experiments on other types of material are in progress. This research is being supported by the Building and Fire Research Laboratory of the National Institute of Standards and Technology under grant no. NANB8D0849.
Copper Side Shields

Marinite Panel

Line Burner

Radiant Panels

Fig. 1: Experimental Setup

Fig. 2: Flame Spread Under Various External Heat Fluxes; Igniter, 18.5 kW/m; No preheat

Fig. 3: Effect of Preheat on Flame Spread; Heat Flux, 5.4 kW/m²; Igniter, 18.5 kW/m

Fig. 4: Effect of Igniter Strength on Flame Spread; Heat Flux, 5.4 kW/m²; No preheat
Experiments have been conducted to study the effects of forced oxidizer flow velocity, grid-generated turbulence, and oxygen concentration on the flow assisted flame spread over a flat solid combustible surface in a ceiling and floor configurations. The tests are conducted with thick PMMA sheets as combustible material, and mixtures of air/nitrogen/oxygen as oxidizer. Flame spread rate, flame length, surface heat flux and products composition are measured for air flow velocities ranging from 0.25 to 4.5 m/sec, turbulence intensities from 1% to 15%, and oxygen mass fractions from 0.19 to 0.4. These parameters address the major heat transfer and chemistry mechanisms that control the spread of the flame. Comparison of the ceiling and floor data provides information about the effect of buoyancy on horizontal flame spread. Most of the experiments at oxygen concentrations different than air are performed under vitiated conditions and in laminar flow, and a few characteristic cases at higher oxygen concentrations and in turbulent flow. The experiments are carried out in a small scale combustion wind tunnel specially designed to conduct flame spread experiments under well characterized oxidizer flow conditions. Although the scale of the wind tunnel is too small to be representative of actual fire situations in buildings, it permits to perform careful parametric studies of the flame spread process, and from them to obtain fundamental information about the controlling mechanisms of flame spread.

It is found that for all oxygen concentrations, and turbulence intensities the flame spread rate increases monotonically with the flow velocity, which is in agreement with flame spread models predictions and is due to the decrease in the flame stand off distance with the flow velocity. For air and ceiling spread, the flow turbulence tends to reduce the spread rate for flow velocities larger than 1 m/sec and to enhance it at lower velocities. For floor spread turbulence always tends to decrease the spread rate. The effect of turbulence on the spread rate appears primarily through the flame length, which decreases as the turbulence intensity is increased. Analysis of the products composition indicates that this effect is due to an enhanced burning of the fuel with the turbulence intensity. These results are affected somewhat by buoyancy effects at low flow velocities. The flow oxygen concentration affects strongly the flame spread rate, which increases with the oxygen concentration. The effect of the oxygen concentration appears both through the surface heat flux and the flame length. The surface heat flux increases with the oxygen concentration primarily because the flame temperature increases. However, the flame length decreases as the oxygen concentration is increased due to a more complete combustion of the vaporized fuel. These effects are more marked under vitiation conditions and low flow
velocities because the gas phase reaction is weaker. The flame length and the surface heat flux exhibit power law correlations with the fuel pyrolyzes length, and the flame spread rate data can be correlated with an expression deduced from a simplified heat transfer analysis of flame spread.

The experimental results indicate that in ceiling spread, buoyancy has two main competing effects. One is the enhancement of the heat transfer from the flame to the solid surface due to the flame being pushed closer to the surface, and the other is the flame quenching by the closer cold wall and boundary layer stratification. For flow velocity larger than 1 m/sec, the enhanced heat transfer is found to be the dominant mechanism and results in a faster flame spread in the ceiling configuration than in the floor. For flows of smaller velocities, the flame quenching effect becomes more important and the opposite result is observed. The species concentration data show that the combustion reaction is less complete in the ceiling than in the floor, and that significant amounts of CO and unburnt hydrocarbons are produced in the ceiling flame spread.

Fig. 1. Comparison of flame spread rate over PMMA in the floor and ceiling geometries.

Fig. 2. Comparison of the exhaust gas composition in ceiling and floor flame spread.
TRANSIENT LAMINAR WIND-AIDED FLAME SPREAD OVER VAPORIZING SOLIDS IN THE CEILING CONFIGURATION

Sanjay Agrawal and Arvind Atreya

Combustion Laboratory
Department of Mechanical Engineering
Michigan State University
East Lansing, MI 48823

(Supported by USDA and NSF)

Wind-aided flame spread is the most hazardous type of flame spread in building fires and has been the most difficult to understand and quantify. An improved understanding of wind-aided flame spread will result in better material hazard classification and greater fire safety.

Our previous experimental investigation has shown that during wind-aided flame spread, the solid-phase undergoes transient pyrolysis while the gas-phase remains quasi-steady. Moreover, the flame length and the flame tip speed are controlled by unsteady pyrolysis of the solid in the burning zone and the rate of progress of the pyrolysis front is controlled by heat transfer from the flame and hot gases to the solid ahead of the pyrolysis front. Recent models have also predicted quantitatively the effects of transient pyrolysis on spread rates.

Previous studies of wind-aided flame spread have primarily focussed on determining the length of the burning zone and have assumed the local pyrolysis rate to be steady. Thus, bypassing the analysis of the pyrolyzing zone. As a result, the pyrolysis front location and its speed, which primarily depends on the ease of ignition of the as-yet-unburned material, have been extensively analyzed. In all of these analyses the surface re-radiation and flame/soot radiation were neglected, and hence the three coordinate system in the preheat zone \((x,y,t)\) could be reduced to two coordinate system (using similarity variables \(\frac{x}{t}, \frac{y}{\sqrt{t}}\)), thus forcing the pyrolysis front speed to be a constant. Our flame spread experiments have suggested that in some cases the pyrolysis front speed accelerates, decelerates or even becomes zero. The first scenario can be attributed to flame radiation and the second to surface heat losses. Inclusion of these in the analysis will result in loss of the similarity and the analysis then demands a numerical solution. Recently Agrawal and Atreya predicted realistic flame front location as a function of steady pyrolysis front speed and time. As far as authors know there has been no study in this particular configuration which predicts both pyrolysis front speed and flame front speed as a function time. This study is an attempt towards this.

Due to transient pyrolysis, the fuel production is significantly smaller than its steady state value. Thus, the diffusion flame moves towards the solid surface because of stoichiometry. This is the primary reason for the flame in the ceiling configuration to lie close to the sample surface and reduces the adverse effect of buoyancy. In the floor or the vertical configurations, the produced fuel itself is buoyantly lifted making the flame lift upwards at lower velocities. This renders ceiling as a preferred configuration for experimentally studying transient wind-aided flame spread. Much lower flow velocities are required in the ceiling configuration than in the floor configuration to suppress buoyant instabilities. Thus, for same length of the combustion tunnel, lower flow velocities provide lower heat flux and more time to study the acceleration or deceleration behavior of the flame front.

During the experiments in the ceiling configuration it was observed that for nearly all the cases tested, the flame lies very close to the sample surface (similar to mm) for the entire flame length. More specifically, the flame stand-off distance is not only small compared to the thermal boundary layer but is also nearly constant. For constant pyrolyzing temperature and constant flame temperature, the streamwise temperature gradient between the flame and the sample is small. Since the gas velocity next to the sample
surface is small, the streamwise transport of energy between the flame and the sample is negligible. Hence, with little error the flame may be assumed to lie on the surface thus simplifying the gas-phase analysis considerably and letting the variables like transient behavior of the solid, surface re-radiation, heat flux shielding due to blowing to be studied in a greater detail.

Simple integral models of gas and solid-phases are developed in the pyrolysis zone which leads into the determination of transient flame front location as a function of pyrolysis front location. It was found in Ref. [2] that blowing of fuel has substantial effect on total fuel production. In the present analysis local blowing correction is used which makes the net surface heat flux into the solid transient (since fuel blowing is transient), though the surface convective heat transfer is still steady. Also with the measured convective surface heat flux in the preheat zone Atreya and Mekki could predict the pyrolysis front speed. In this paper we use the correlation for surface convective heat flux in the preheat zone as suggested in Ref. [9] and use surface re-radiative loss and the steady gas-phase and transient solid-phase models of Agrawal and Atreya to predict both the flame front location and pyrolysis front location as a function of time, free stream velocity, oxygen mass fraction, surface emissivity and external heat flux. Also the maximum sample length is predicted that a non-radiating flame can travel. Future considerations are to include simplified flame radiation in order to predict acceleratory spread rates and also analyze charring solid in the same configuration.

REFERENCES

Calculating Flame Spread on Horizontal and Vertical Surfaces

Gamal N. Ahmed¹, Mark A. Dietenberger² and Walter W. Jones

Building and Fire Research Laboratory
National Institute of Standards and Technology

The flame spread model described in this paper is a new algorithm which provides the capability to calculate a self-consistent fire based only on bench scale fire data. This model simulates object fire growth and burnout of a slab in a room and produces acceptable predictions of the spread of fire, smoke and production of both toxic and nontoxic gases. Thus the simulation allows a fire to grow realistically, possibly making a hole in the material surface. The algorithm uses empirical data, gathered from standard test apparatus, including the Cone Calorimeter and the LIFT (lateral ignition flame spread test method). By basing the transient pyrolysis description on test methods, we avoid the practical difficulties associated with an explicit calculation of radiation blocking and material charring. The intent of the project was to develop an algorithm which could be utilized in a complete model of a fire in a building. The three-dimensional aspects of the flame spread model include: first, panels made of combustible materials with different thicknesses and at various orientations; second, flames of two basic types, pool fire and purely wall fire. The former has a flame spreading polygon on a horizontal panel. The latter is used either for inclined or vertical panels; third, a radiation heat exchange between objects, flames, and gases.

The first release of the Fire Hazard Methodology in 1989 was a prototype. Its focus was to incorporate the research which had already been done at NIST and elsewhere into a relatively small reference guide, along with the computer software which implemented this work. An example of such an approach is the requirement for a specified fuel. In essence, the data used for the fire specification comes from free-burn data obtained in a furniture calorimeter or similar device. The fire can be constrained by a lack of oxygen, but is not affected by radiative feedback. Since then, the Building and Fire Research Laboratory has directed much of its research into improving and enhancing this product.

The FAST (fire and smoke transport) model utilizes the distinct gas zones concept as a compromise between a network model and a finite difference model. The basic predictive equations are derived from the laws of conservation of the mass, momentum and energy. There are usually two zones, or control volumes, per compartment. The two zones are referred to as upper and lower respectively. Transfer can occur from zone to zone or compartment to compartment in a fire driven environment. These equations are written so that the actual physical phenomena which affect the environment are couched as source terms and appear on the right-hand side. The flame spread model (FSM) discussed

¹ Fire Research Engineer, Portland Cement Association (PCA), Skokie, Illinois

² Research Engineer, Forest Products Laboratory, Madison, Wisconsin.
in this paper is couched simply as another algorithm. Thus, the FSM provides source terms for the radiative heat transfer, plume mass rate and heat and mass release rates. The numerical solution of these equations has been shown to be both practical and sufficiently detailed.

The flame spread model has been applied to simulate a burning panel of HO/FRPU (heavy Olefin foam material/fire retardant polyurethane) in a single compartment where the fire has been specified as one of the constrained fires. The simulated panels have different orientations (horizontal panel and vertical walls), different geometries with different thicknesses, and different fire positions in the room of fire origin (center of the room and along a wall of the room) with different ignition locations on the panels. In these applications, the effect of door open (width and height of the door of the room of fire origin) on the flame spread rate developed on the panel, the pyrolysis rate, and the upper hot gas layer depth has been demonstrated and analyzed. For full size wall, the simulated wall has slightly smaller width and height (3.8 x 2.9 m) than the width and height of the room (4.0 x 3.0 m) to avoid some difficulties in shape factor and heat exchange area calculations for the transfer of heat by radiation.

The flame spread on a horizontal 1.0 m² panel initially proceeded at a minimum rate and then accelerated toward the panel edges as a result of radiative heating from the fire plume and the growing upper gas layer. Once the flame spread contours completely cover the panel, the fire experienced its greatest growth rate as indicated by the heat and mass release rates. Eventually, the fire growth reaches a limit dictated either by the flame soot volume fraction reaching its maximum value for the given flame base radius or by the burnout of some surface elements. While the panel surface is burning, the flame spread contours begin to recede into the cushion thickness until the whole panel material is burned out.

Vertical flame spread from a piloted ignition source moves up rapidly until it reaches the top of the wall, and then moves sideways at the top in an accelerated fashion due to radiative heating from the growing upper gas layer. The later deceleration of the flame spread is due to much lowered preheating of virgin elements below the upper gas layer. Changing the door opening has negligible effect on the predicted flame spread. However, the door opening has a pronounced effect on the upper gas layer depth. The lower the top of the door or narrower the door opening is, the higher is the pyrolysis rate of the burning wall. The maximum depth of the hot gas layer approximately coincides with the peak in the pyrolysis release of the burning wall material. Varying door opening, both the hot gas layer depth and the pyrolysis rate of each case have reached their limiting values, which is an important modeling feature.

CRITICAL CONDITIONS FOR FLAME SPREAD AND A RATIONAL FIRE HAZARD MATERIAL CLASSIFICATION METHOD

by

Michael A. Delichatsios, Y. Chen and Mary M. Delichatsios

The fire hazard of a material can be characterized by two parameters: a maximum flame spread length scale, $L_m$, and a total spread time $\tau_t$. These parameters, defined below, include also the effects of the transient pyrolysis history of a material which for non-charring materials is determined by the ratio $L/(C_\lambda \Delta T)$ (1). For charring materials a modified effective heat of pyrolysis is introduced, 

$$\Delta H_p^{eq} = \Delta H_p / ((1+\lambda) d_c / \delta_v)$$

where $\lambda$ is defined above, $d_c$ is a char conductance length and $\delta_v$ is a thermal length.

The maximum flame spread length scale parameter was derived starting from the general equations for upward flame spread, using fundamental properties and parameters affecting upward fire spread, such as thermal properties of the materials, flame height correlations and flame heat flux measurements. From this analysis,

$$L_m = 1.4 \times 10^{-4} \left( \frac{q''_{net} x_A \Delta H}{\alpha} \right)^2$$

where $q''_{net} = q'' - \alpha \Delta T$, $x_A$ is the combustion efficiency and $q''$ is the total heat flux to the wall from the flames and/or other sources. The total spread time, proportional to the pyrolysis time, depends (for thermally thick conditions) on the material thermal inertia, the pyrolysis temperature and the total heat flux from the flames and/or other sources to the wall, i.e.,

$$\tau_t = 5 k \rho c \Delta T (\rho^{1/2} \alpha^{2/3}) / q''_{net}.$$  

The maximum flame spread length scale has a physical interpretation, i.e., it is the actual maximum flame spread height, for values of $L_m < 1.8$ m, when the flame heat flux can be assumed approximately constant, equal to 25 kW/m$^2$. For larger values of $L_m$, the radiative component of the flames increases substantially and the value of $L_m$ given by Eq. (1) no longer has physical meaning. But even for this case, it can be used, for a relative fire hazard classification of materials if one includes the change in total heat flux (see Fig. 2). Similarly, the total spread time, $\tau_t$, is the time required for the flames to reach a height equal to 90% of the maximum flame spread length. Obviously, the total spread time, $\tau_t$, has a physical interpretation as long as $L_m$ has physical meaning, i.e., for $L_m < 1.8$ m. Figures 1 and 2 show the maximum flame spread length scale, $L_m$, and the total spread time, $\tau_t$, from Eqs. (1) and (2) for two values of total heat flux to the wall, 25 kW/m$^2$ and 40 kW/m$^2$. The material properties were taken from the literature.

As can be seen from these figures, the maximum flame spread length scale can be a good predictor of the relative behavior of upward fire spread of the different materials in small fires (25 kW/m$^2$) and in larger fires (40 kW/m$^2$). For example, polyethylene (PE), PE/PVC cables, polystyrene (PS) are materials on which upward fire spread is not fast and fire only spreads up to heights of less than one meter if the only heat flux to the wall is the heat flux from
the flames ($Q_f^* = 25 \text{ kW/m}^2$). For the same conditions, (i.e. $Q_f^* \leq 25 \text{ kW/m}^2$) particle board and rigid foam are non-spreading materials ($\tau_{m} < 20 \text{ cm}$), whereas PMMA, PVC and polypropylene (PP) are fast spreading materials.

Figure 2 shows a similar relative behavior of upward fire spread of different materials in larger fires where the total heat flux to the wall material is 40 kW/m$^2$. A larger fire could result from external heat fluxes, a large ignition source (larger than 1 m), geometric configuration (i.e., a corner) or a high value of $\tau_{m}$ ($\tau_{m} > 1.8 \text{ m}$) as in PMMA, PVC and Polypropylene (PP). As can be seen from this Figure, the relative behavior of the different materials, i.e., relative magnitude of $\tau_{m}$, concerning upward fire spread in large fires, is not necessarily the same as it is in small fires. For example, it is worthwhile to notice that polyethylene (PE) behaves as a slow-spreading material in a small fire but as a fast spreading material (similar to PMMA) if the fire develops to a larger, more intense fire as a result of external heat fluxes and/or a large ignition source (larger than 1 m).

REFERENCES


Figures 1, 2. Maximum Flame Spread Length Scale, $\tau_{m}$, and Total Spread Time, $\tau_{t}$, for different materials for two flame heat fluxes to the wall: $Q_f^* = 25 \text{ kW/m}^2$ and $Q_f^* = 40 \text{ kW/m}^2$, respectively.
Fire Resistant Polymer Grafts

Marc R. Nyden and James E. Brown
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899
and
Barry Bauer
Materials Science and Engineering Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

ABSTRACT

Flammability is a major concern when polymeric materials are used in buildings, aircraft and ships. This realization has provided the impetus for research into the synthesis of new and more fire resistant polymers.

Figure 1. The thermal degradation of linear polyethylene interacting with a surface (white zig-zag lines). Carbon atoms involved in cross-links are highlighted in white.
Research conducted at BFRL and MSEL has focused on finding ways to increase the tendency of plastics to char when they are burned. There is a strong correlation between char yield and fire resistance. This follows because char is always formed at the expense of combustible gases and because it protects the underlying material from the heat generated in gas phase combustion. Factors which promote the formation of char have been identified in molecular dynamics simulations of degrading polymers. The scene depicted in Figure 1 is representative of what happens when linear polyethylene and related polymers burn. The chains fragment into fuel before a significant number of cross-links can form. An incipient char, such as the one depicted in Figure 2, however, was produced when hydrogens were removed from the model polymers at the onset of the simulations. In this way, cross-linking reactions are given a head-start so that they can effectively compete with fragmentation. The prediction that cross-linked polymers undergo further cross-linking when burned and eventually form high molecular weight, thermally stable chars has been confirmed in Cone Calorimeter flammability measurements made on both γ and β-irradiated polyethylene and on chemically cross-linked poly(methyl methacrylate).

![Figure 2. The thermal degradation of cross-linked polyethylene.](image)

The behavior exhibited by the β - irradiated polyethylene when it was burned was particularly striking. The more highly cross-linked layer on the top of the sample, charred and formed a large bubble indicating retention of the gases generated in the degradation of the interior. Eventually, the bubble broke and the sample ignited; leaving behind a thin carbonaceous skin. This experiment suggests that it may be possible to achieve significant reductions in flammability by grafting fire resistant shells on to the surface of processed plastics. Experiments with new materials, synthesized by grafting vinylidene chloride and other fire resistant monomers, to the surface of polystyrene are underway. The effect of the thickness and chemical nature of the graft on the relative flammabilities of these composites will be examined.
ACOUSTIC EMISSION OF STRUCTURAL MATERIALS EXPOSED TO OPEN FLAMES

William Grosshandler and Margaret Jackson
Building and Fire Research Laboratory
National Institute of Standards and Technology

The objective of this experimental study is to assess the viability of using the acoustic signal emitted by a variety of structural materials exposed to nonuniform heating as an early indicator of a fire. Beams of aluminum, gypsum, plywood, poly-methylmethacrylate and poly-vinylchloride, simply supported across a 0.5 m span, were subjected to a 0.9 kW open flame. Sensors were mounted directly on the beams, 0.3 m from the center of the flame, to record high frequency acoustic emissions. The measured signals varied in power and in number with the material and thickness of the specimen.

The acoustic emission (AE) from a bulk material is associated with an abrupt change in stress following a microscopic dislocation. The stress relief generates an elastic wave which radiates throughout the material at the speed of sound. The thermal energy released by the flame impinging on the beam, plus the combustion of the material itself, creates a temperature gradient, and leads to thermal stresses in the material. A piezoelectric transducer located remotely from source can detect the elastic wave created by the stress relief just milliseconds after it occurs. The strength of the acoustic source depends upon the suddenness with which the stress is relieved. For an instantaneous release, the source acts a dipole whose strength increases directly with the thermal diffusivity, elastic constant, and thermal energy density, and inversely with the thermal conductivity. The number of AE events will increase with the number of defects or discontinuities in the material; hence, a composite material is likely to be more active than a pure substance.

The samples varied between 3 and 18 mm in thickness. The heat flux was maintained approximately constant over a circular area about 50 mm in diameter, and the time of exposure to the flame was adjusted between one and six minutes, depending on the temperature rise in the material, to prevent the AE sensors from overheating. The maximum temperature measured 0.1 m from the flame was 106°C in the 2 mm thick aluminum sample, compared to only 36°C for the 12 mm gypsum board at the same location. In the latter case, the increase in temperature over the ambient was 0.8 °C at the location of the AE sensors, 0.30 m removed from the flame.

An acoustic emission event can be described by its rise time, amplitude, frequency, duration, and energy. The main distinguishing feature among the materials investigated was the event duration, which averaged just under a millisecond for the 12 mm thick aluminum beam and 0.15 ms for the gypsum board, the difference being attributable to the internal damping properties of the two materials.

The number of AE events in a minute and the accumulated energy release during the heating cycle provide a good measure of the overheated state of some structural materials. The 12 mm thick plywood was particularly susceptible to acoustic emission, with about 43 events/minute being recorded. The gypsum board produced 16 events/minute and the 12 mm aluminum just 0.5 events/minute. The thinnest aluminum plate did not respond above the background level (0.3 events/minute) even though it reached the highest temperature.
The differences in cumulative energy were even more striking, with the plywood being four times more energetic than the gypsum board even though the heating period for the wood was half as long, and 30 times more energetic than the aluminum.

The major conclusions to come out of this study can be summarized as follows:

1. Some common structural materials emit high frequency acoustic signals when exposed to a modest flame.

2. Composite materials exhibit much higher AE activity than homogeneous materials.

3. The use of AE transducers as early sensors of hidden structural fires remains a viable concept.

4. Critical issues remain to be investigated if this technique is to be useful for fire detection; e.g., false signal discrimination must be verified, the maximum practical area covered per sensor must be determined, and durability and reliability must be examined.
MACHINE VISION FIRE DETECTION

Douglas B. Schwartz

Air Force Civil Engineering Support Agency
Tyndall Air Force Base, Florida

A new type of fire detector under development "sees" a fire visually and makes a reliable decision based on known visual characteristics of flames. The Machine Vision Fire Detection System, currently in the engineering prototype stage, has already demonstrated extremely high accuracy in discriminating actual fires from false alarm sources, promising virtually 100% reliability at very high speed and reasonable cost. By dramatically reducing the false alarm rate, this technology promises to save the Air Force alone over $3 million per year in elimination of unnecessary releases of firefighting agent, while enabling improved fire protection of critical Air Force assets.

Current technology fire detectors sense smoke, heat, or electromagnetic energy such as ultraviolet (UV) or infrared (IR) emissions. The latter type, optical fire detectors (OFDs) are used when fast, remote sensing is required. These sense electromagnetic emissions in particular bands emitted by hydrocarbon fires, specifically the 0.18-0.24 micron band for UV and the 4.4 micron CO2 band for IR. The disadvantage of this is that any emitter at these frequencies will give an indication of a fire. Ultraviolet detectors, for example, are commonly set off by reflected sunlight and arc welding. Infrared detectors can be set off by hot exhaust manifolds on aircraft ground equipment, propane torches, and other heat sources. Reliability of these detectors has been improved by multispectral detectors, including UV and IR and dual IR.

Certain fundamental limitations remain, though. Optical fire detectors trade off speed for accuracy; the faster the system is set to detect a fire, the higher the false alarm rate. Typically, optical detectors are slowed to 3-30 seconds detection speed from a theoretical speed of less than 1/100 second. OFDs cannot judge the magnitude of a fire; a small fire close up emits the same energy to the detector as a large fire further away. Because of the inverse square law, maximum reliable range of OFDs is about 120 feet, a serious limitation in large spaces such as warehouses.

Machine vision fire detection seeks to overcome these limitations by characterizing the fire in the visible spectrum, using similar criteria that humans use to tell fires from non-fires. Detection begins with a solid-state video camera, which uses a CCD (charged-coupled device) to convert light into electronic information. The CCD is a square silicon chip, typically made up of 512 picture element, or pixels, on a side. Each pixel transmits the intensity of red, green, and blue light in a digital signal, which is transmitted into computer memory in a sequential scan.

The image is first checked for bright areas (intensity above a certain threshold). These areas are then "boxed and checked for color within red, green, and blue parameters. The image is then compared to the next frame, taken 1/30 of a second later.
This will reveal changes in color from frame to frame, behavior of the edge of the object, and growth rate. Actual fires exhibit rapid color changes from frame to frame, have highly variable edges, and tend to grow outward from a starting point.

Size of the fire is computed by counting the "fire" pixels. For a fixed installation, the system will be calibrated at installation to relate position within its field of view to a particular size. Thereafter, the pixels across the base of the fire can be summed and size computed from the number of pixels from the lower edge of the field of view. Portable or mobile installations will use two cameras on a known baseline for range estimation.

Since actual size and growth rate are known, the relative threat of the fire can be determined, allowing a flexible response. For example, the machine vision detector might be linked to an automatic suppression system capable of dispensing tens of thousands of pounds of firefighting foam onto a hangar floor. This massive response would be overkill against a small, steady-state event such as a oil rag fire. In this case, an alarm would be sounded to alert personnel, who could suppress the fire with hand extinguishers. If the fire exceeded a certain size, or if growth rate became high, the suppression system would be activated.

Hardware components of the system, including the video camera, image processing hardware, and microprocessor, are all available "off the shelf." This drives down cost and risk. Current cost of the components is about $2500 and is expected to drop. Time to make the fire/no fire decision is 1/10 second; if necessary, this can be reduced tenfold through use of a faster processor.

The machine vision fire detector is currently in prototype stage and has undergone periodic field tests against actual fires and false alarm sources. At the end of 1992, final prototypes will undergo full scale validation. The system is expected to be incorporated into a major Air Force upgrade of hangar fire protection systems. The technology will also be applied to development of portable fire detection and suppression systems and on-board aircraft fire protection.
DETECTION OF UNDERGROUND MINE FIRES

By Ronald S. Conti

U.S. Department of the Interior, Bureau of Mines, Pittsburgh Research Center
P.O. Box 18070, Cochrans Mill Road, Pittsburgh, Pennsylvania 15236

ABSTRACT

An underground mine fire not only is a serious threat to life and property, but also results in economic losses such as injuries, mining equipment, coal reserves and production, and jobs. From 1983 to 1988, 86 underground coal mine fires were reported to the Mine Safety and Health Administration (MSHA) (1). These fires caused 29 fatalities, numerous injuries, the sealing of several mines, and financial losses totaling tens of millions of dollars. During the incipient stage of a fire, smoke and gaseous products including CO are produced and released into the mine atmosphere. If not detected in the early stages of combustion, these fires can spread rapidly and are both difficult and dangerous to extinguish. Also, if they are not contained in the early stages, portions of a mine or the entire mine must often be sealed for long periods.

This U.S. Bureau of Mines study describes the results of research conducted in the Experimental Mine at Lake Lynn Laboratory (2) on the response of fire-sensors to simulated mine fires, which include: a slowly-developing coal/conveyor belt fire, a rapidly burning liquid-fuel belt fire, and a liquid-fuel belt fire in the presence of diesel exhaust. The fire detection studies (3) to be discussed were conducted in A-drift. A detailed layout of a typical underground fire and detection scenario is shown in the perspective view in figure 1. During the experiments the airflow of the mine is reversed, so that the combustion products are exhausted through the main fan. The moveable bulkhead door in D-drift is closed and the bulkhead door in E-drift is in an open position. Temporary stoppings are installed at the last crosscuts of B and C-drifts. The airflow can be adjusted with one of the four positions of the main fan and a 0.61 m butterfly valve located in the bulkhead door of D-drift. The airflow is monitored with a vane-type anemometer 15.2 m inby the fire zone.

Products of combustion detectors were used at two positions along the mine entry. Two pump operated, electrochemical CO sensors (see figure 1, detail I) are mounted 15.2 m inby the fire zone in the middle of the entry. One CO sensor is mounted near the roof and is labeled CO-50-Roof. The other CO sensor is mounted directly below, 0.66 m from the floor and is labeled CO-50-Mid.

Two thermal-contact detectors were located near the fire source. A chromium-aluminum thermocouple is placed at the roof 0.3 m inby the fire zone and a continuous length of a thermal line-type fire detector (heat sensitive cable) is mounted at the roof from the portal of A-drift and extends 30.5 m past the fire zone. A line-type fire detector is a twisted pair of insulated wires that short circuit when exposed to temperatures in excess of 68° C (155° F).

Five fire sensors were mounted as shown in detail II of figure 1, in the entry cross-section at a point 274 meters (900 ft) inby the fire zone. Three diffusion-type electrochemical CO sensors are used. Two are mounted at the roof, labeled CO-Roof and CO-Roof-B, and represent two brands of CO sensors. The other CO sensor is mounted 0.66 m from the floor on the rib, identified as CO-Rib, and is of the same brand as the CO-Roof sensor. A commercially-available ionization-type smoke sensor, labeled smoke, was mounted on the rib, with the intake sampling point located beside the CO-Roof-B sensor at the 274 m location. A prototype diesel discriminating detector (4), labeled DDD, was mounted beside the intake sampling point of the Smoke and CO-Roof-B sensors at the 274 m location. Carbon monoxide and smoke sensors are calibrated before each tests.
During these tests, several mine fire sensors were evaluated with respect to sensor placement, spacing, and type. The data indicate that smoke sensors will alarm several minutes before carbon monoxide (CO) sensors; and that, in the presence of diesel exhaust, a prototype diesel-discriminating smoke sensor can successfully function without being sensitive to the diesel contaminants. The vertical placement of sensors in the entry near the fire was also shown to be critical in terms of alarm times. Additional data showed that variations exist in response time and level of response for two types of electrochemical carbon monoxide sensors. Results also indicate that early detection of fires will improve the probability of miners' escape because of reduced smoke concentrations during the incipient stages of the fire.

***REFERENCES***


Figure 1.---Perspective view of underground fire detection scenario in A-drift. Exploded views of sensor placement with respect to fires are shown in detail I and II.
FLOW THROUGH HORIZONTAL VENTS IN ENCLOSURE FIRES

Y. Jaluria and Q. Tan
Department of Mechanical and Aerospace Engineering
Rutgers University, New Brunswick, NJ 08903

Introduction

Buoyancy induced flows in enclosures, with respect to the transport induced by fires in rooms and other compartments, have received a considerable amount of attention in the literature. However, not much work has been done on the flow through openings or vents such as those between connecting rooms in buildings and between quarters in ships. The work done on the flow through horizontal vents, such as the one shown in Fig. 1, is very limited. The flow rate can be estimated, as done for vertical vents, by using Bernoulli’s equation (Emmons, 1988 and Hinkley, 1986). However, this model breaks down, over certain ranges of the pressure and density differences, for problems where both density and pressure differences exist across the vent (Cooper, 1989).

Some experimental work has been done on the buoyancy driven flow through horizontal vents, particularly for the special circumstance of pressure difference across the vent $\Delta p=0$ (Epstein, 1988). Mercer and Thompson (1975) studied the flow between two fluid regions connected by an inclined circular pipe. Conditions for purging of the density induced flow were investigated. A total purging or flooding implies a unidirectional flow through the vent from one region to the other.

The standard vent-flow model, based on Bernoulli’s equation, assumes unidirectional flow and breaks down if a bi-directional flow exchange occurs under the combined effects of buoyancy and pressure. This model predicts zero flow when $\Delta p=0$. The volume flow rate $Q_u$ at the vent into the upper region from the lower region is taken as uniform and given by the model as $Q_u = C_D A_V \sqrt{2 \Delta p / \rho_l}$, where $C_D$ is the flow discharge coefficient, $A_V$ is the vent area of cross-section and $\rho_l$ is the density of the fluid in the lower region (Heskestad and Spaulding, 1989).

Similarly, if the pressure difference is reversed, the flow $Q_D$ is obtained. However, there is a flow across the vent due to the density difference $\Delta p$, even if the pressure difference $\Delta p$ is zero, due to the fluid in the upper region being heavier than that in the lower region. Cooper (1989) has developed an analytical model, employing an exchange-flow component and has determined the steady-state rate-of-burning in a ceiling-vented room, such as the one sketched in Fig. 1.

Experimental Arrangement and Results

This paper is directed at the flow through horizontal vents for nonzero pressure and density differences across the opening. An experimental system based on a fresh-water/saline-water flow arrangement is employed. The density difference ratio $\Delta \rho / \rho$ is varied from 0.0 to about 0.2, in order to cover the range encountered in typical vented-compartment fires. The salinity is measured by means of a hydrometer. The pressure difference $\Delta p$ across the vent of diameter $D$ is obtained by keeping the upper region open to the atmosphere and pressurizing the bottom region. The volume flow rate across the vent is obtained by measuring the dilution of the brine solution contained in the upper region and the net flow rate $Q_o$, as discussed by Tan and Jaluria (1992).

Several interesting and important results have been obtained. Using a shadowgraph for flow visualization, a bidirectional flow was found to arise across the vent at $\Delta p=0$, with denser fluid descending in the central portion of the opening and lighter fluid rising in the outer portion. As the opposing pressure difference $\Delta p$ was increased, this downward flow was found to decrease, ultimately giving rise to a unidirectional upward flow. This purging, or flooding, pressure difference $\Delta p_e$ is of order $(g \Delta \rho L)$ for a vent of height $L$, $g$ being the gravitational acceleration. Some typical results on the flooding pressure difference $\Delta p_e$ are shown in Fig. 2. The measured values are in the range that may be derived on the basis of scale analysis. The results were found to be repeatable, within a few percent, of the measured values. However, the transition from bidirectional to unidirectional flow was found to be fairly gradual, indicating the downflow $Q_D$ to gradually reduce to zero as the pressure difference $\Delta p$ is increased. With increasing values of $(g \Delta \rho D)$, at a given $L/D$, the purging pressure difference $\Delta p_e$ was found to increase, as expected.

The net upward flow rate $Q_u$ was also measured and the other flow rates $Q_o$ and $Q_D$ determined from the measurements of density and overflowing fluid in the upper region. Figure 3 shows some of the characteristic results obtained. At zero $\Delta p$, the upward and downward flow rates are equal, yielding a zero net upward flow. As the pressure difference increases, the flow rate $Q_o$ increases, with the effect of the density difference $\Delta p$ diminishing as
Ap becomes large. Also, at very low pressure differences, the density difference effects dominate the flow across the vent. Thus, clearly the basic trends indicated by Cooper (1989) are obtained.

The results may also be given in terms of a flow discharge coefficient \( C_D \) as defined earlier. This discharge coefficient is around 0.61 for an orifice (Emmons, 1988), if buoyancy effects are neglected. However, as seen in this work, the flow rates decrease if the opposing buoyancy effects are increased. Therefore, the discharge coefficient may be correlated in terms of the buoyancy parameter \( B \) and \( L/D \), where \( B = (g \Delta \rho D)/\Delta p \). A correlation obtained from the data over the parametric ranges considered is of the form:

\[
C_D = 0.61 + 0.07974B^a(L/D)^b
\]

where the constants \( a \) and \( b \) were obtained as -0.2246 and 0.0278, respectively. As \( B \) approaches zero, the value of \( C_D \) approaches 0.61. Also, the flow rate decreases as \( B \) increases. This result applies as long as the buoyancy effect is not predominant, or the pressure difference is not close to zero, i.e., \( B \to \infty \). For that case, the earlier results at \( \Delta p \to 0 \) may be employed.

Acknowledgments

The authors acknowledge the financial support provided by the National Institute of Standards and Technology, under Grant No. 60NANB7D0745, for this work and the several discussions with Dr. Leonard Y. Cooper on this problem. The help of Mr. R. Chiruvella in preparation of this manuscript and in obtaining some of the correlating equations is also acknowledged.

References


Fig. 1. Sketch of the fire in a room with ceiling vent.

Fig. 2. Measured purging pressure difference \( \Delta P_e \) as a function of the buoyancy effect.

Fig. 3. Measured Volume flow rates for varying \( Ap \) at \( L/D = 0.25 \).
Experimental Study of the Exchange Flow Through a Horizontal Ceiling Vent in Atrium Fires

Tokiyoshi YAMADA
Fire Research Institute
Fire Defense Agency, Ministry of Home Affairs
Mitaka, Tokyo, JAPAN

Leonard Y. COOPER
National Institute of Standards and Technology
U.S. Department of Commerce
Gaithersburg MD, USA

Introduction. The object of this work is to study experimentally flow rates through a horizontal top vent in an atrium-type compartment with a fire and under relatively low cross-vent pressure differences. The results will be used to develop a comprehensive atrium fire model. From the point of designing atria for life safety (e.g., insuring safe evacuation from all threatened spaces) reliable estimates of the flow through top vents are important. For example, outward smoke movement generally leads to reduction of smoke layer temperature and thickness and inward flow of cool outside air enhances potentially hazardous mixing of the smoke through the entire atrium space.

The physical phenomena have been studied by Epstein[1,2] and Jaluria[3] with salt-water experiments. Also, limited experimental work with hot/cool gas exchanges were carried out by Tu [4] who used an ethyl alcohol burner heat source. Based on Epstein's experimental result, Cooper [5, 6] has been developing a submodel for this kind of phenomena suitable for use in two-layer zone-type fire models. However, available data to verify and advance this kind of submodel are not adequate. The present experiments are designed to acquire required data and insights on the exchange flow process. This effort, which uses a reduced scale model with a gas burner, is similar to the liquid-fuel burner experiments of Tu [4].

Reduced-Scale Experiments. Reduced scale tests were conducted to study the exchange flow rate through circular, horizontal, top-centered vents (diameters, 0.3 m, 0.2 m, and 0.1 m). A cubic test compartment (0.8 m side) was constructed of 0.05 m (0.0125 m x ceiling) thick ceramic fibre board. Fig. 1 is a sketch of the test apparatus. A 0.15 m diameter natural gas (11,000 Kcal/Nm³) burner was located flush with the floor and at the center of the enclosure. Heat release rate (0.8 - 5.6 kW) was controlled by a mass flow controller. Inside the test compartment was a thermocouple tree (type K with 0.127 mm beads) with 17 measuring points to measure gas temperature versus height. Using a vertical sliding probe, gas volume concentration was measured at 0.01 m, 0.40 m and 0.79 m above the floor. For future experiments, the test fixture was designed with a capability for specified uniform-flux air ventilation at the floor surface. Similar preliminary tests were carried out in a smaller test compartment (0.43 m side, the same size as was used in [4]) and burner (0.1 m diameter), but without any ventilation capability. To allow for visual observation within the compartment, both the large and small test fixtures (referred to below as large and small scale) were outfitted with a floor-to-ceiling glass strip in the center of one wall. During some tests, flow visualization was carried out with the use of smoke generating pellets placed outside the compartment and near the vent opening.

Estimation of Flow Rate. Direct measurement of the bi-directional flow through the vent is not possible. In this experiment, the exchange flow rate was estimated by using the oxygen consumption method. We suppose that heat release rate calculated from mass burning rate is nearly equal to the estimated energy release rate obtained by oxygen consumption method. Neglecting the volumetric flow rate of the fuel compared to that of the vent inlet flow, V_amb, and for steady state this leads to

\[ \dot{V}_{amb} = \frac{Q_f}{H \mu_{amb} \rho_{amb} (1 - \mu_{bot} / \mu_{amb})} \dot{V}_{bot} = \frac{\rho_{amb} \dot{V}_{amb}}{\rho_{bot} \dot{V}_{bot}} \]  

where \( \mu \) is O mass fraction, \( H \) is specific energy (13.1 MJ/kg), \( Q_f \) is the fire's heat release rate, the subscripts amb and bot refer to conditions above and below the vent, respectively, or to the vent flows issuing from outside (ambient) to inside or from inside to outside the test fixture, respectively.

\[ \dot{V}^* = \frac{V}{\sqrt{D^2 g \Delta p / \rho}} \]

where \( D \) is vent diameter, \( g \) is acceleration of gravity, \( \rho \) is density, \( \rho = (\rho_{amb} + \rho_{bot}) / 2 \) and \( \Delta p = (\rho_{amb} - \rho_{bot}) \). Experiments of [1] in a top-vented compartment with geometry similar to ours resulted in an equation correlating \( \dot{V}_{amb}^* \) vs \( L/D \), where \( L \) is vent depth. The correlation is plotted in Fig. 2. For relatively small-L/D, \( \dot{V}_{amb}^* \) is 0.055. For experiments with incompressible fluids (e.g., fresh-water/saltwater), \( \dot{V}_{amb}^* = \dot{V}_{bot}^* \) and the cross-vent pressure difference is zero. As indicated by the second of Eq. (1), this is not generally true for steady heated-air experiments.

![Fig. 1 Sketch of experiment; dimensions in mm](image-url)
Included in Fig. 2 are data from [1], [3], and [4]. The present data are listed in Table 1. The large scale data were at steady state, at least 2 hours after ignition. The small scale data were at quasi-steady-state, about 1/2 hour after increases in fuel flow rate. The data, reduced using Eq. (1), are included in Fig. 2. In the latter data reduction the \( \psi_{x0} \) of Eq. (1) was estimated by the measured near-ceiling \( Q \) volume fraction ratio, in the case of large scale, and by the average floor-to-ceiling \( Q \) volume fraction ratio, in the case of small scale. In both cases, \( \psi_{x0} \) was estimated from near-ceiling time-averaged temperature measurements. Data is included in the table only for those tests where \( Q \) concentrations near the ceiling were less that 0.19. This insured estimates of \( (1 - \psi_{x0})/\psi_{amb} \) in Eq. (1) with expected errors of less than 25 percent.

**Discussion.** The small-scale data and the data of [4] compare reasonably well with the correlation of [1]. Some discrepancies with data from incompressible fluid experiments are to be expected because of the effects of large cross-vent density differences and non-zero cross-vent pressures. It is important to take these effects into account in a mathematical model of the exchange flow phenomena.

The large-scale data do not correlate well with the small-scale data. One possible explanation for this, which has not yet been confirmed, is that leakage through the joints of the large fixture plays a significant role in the present, relatively-small, cross-vent pressure experiments. Leakage effects could be significant in the large scale fixture and not in the small scale fixture because of 1) its potentially-leaky floor-ventilation design feature, 2) its larger surface area, and 3) the larger cross-wall/floor pressure differences (i.e., at comparable inside temperatures). This will be investigated prior to future tests.

In the tests with the 0.1 m vent the burner flame always exhibited extended, laminar-like behavior. This was different from all other tests where the flame and its plume exhibited turbulent-like behavior. The former laminar characteristic of the flow persisted even near the elevation of the vent itself. Therefore, it is no surprise that in Fig. 2 these data do not correlate well with the Epstein data, which involved turbulent-like flows near the vent. (For practical interest, the objective of the present study is to understand the exchange flow through "real" vents under expected turbulent flow conditions). The 0.1 m "blocked" flame experiments involved tests where the axis of the laminar flame was blocked with a small metal plate, close to the burner surface, in an attempt to stimulate turbulent-like behavior in the plume near the vent elevation. This was partially successful. By simply blocking the flame, there was a significant change in the estimated vent flow conditions; the "blocked" data are definitely closer to the "turbulent-flow" correlation.

**References**

A Model of Combustion and Convection in Enclosures of Complex Geometry

Kevin B. McGrattan, Ronald G. Rehm and Howard R. Baum
National Institute of Standards and Technology
Gaithersburg, MD 20899

We present a model describing diffusion-controlled combustion and convection in 2D enclosures of general polygonal shape at high Reynolds numbers. The model is intended to simulate the transport processes associated with the movement of smoke and hot gases in a fire. The combustion model can be described as follows. Lagrangian particles representing the fuel are introduced into the flow at the prescribed location of the fire. Each particle is active, acting as a heat source which induces buoyant convection. The trajectory of each particle is determined as part of the solution of the large scale convection problem. This is the high Reynolds number limit of any diffusion-controlled combustion originating on the surface of discrete fuel parcels. The particles may also serve as a visualization aid, allowing one to view the evolution of the flow field in time by following the trajectories of the particles, color-coded to indicate the temperature of the fluid.

The hydrodynamics are based on the time-dependent Navier-Stokes equations in the Boussinesq limit [1]. Viscous dissipation and thermal conduction are included in the model, but no turbulence model or other empirical parameters are introduced. In the two-dimensional case, we rely on the high resolution of the computational mesh to achieve Reynolds numbers on the order of $10^5$, which correspond to full-scale in small room fire scenarios. To accommodate the irregular shape of the enclosures, the physical domain is mapped conformally onto a rectangle with the numerical Schwarz-Christoffel mapping package SCPACK. The transformed equations are then solved by finite-difference techniques [2]. Typical computations are performed on grids ranging from a quarter to half a million grid cells, permitting Reynolds numbers of the size described above. This approach allows us to consider room fire phenomena in realistic geometries, including the flow out of a window, combustion in a corridor with a door, and the flow generated in a representative segment of a building.

The result of a typical computation is displayed on the following page. Here a light fluid representing smoke and hot gases inside the small enclosure mixes with a heavier fluid representing fresh air outside. The Reynolds number for the computation is 40,000, the grid size is $1024 \times 256$. No-slip, adiabatic boundary conditions have been imposed at the walls. Computations similar to the one presented here have been compared with salt water experiments performed at the University of California at Berkeley by Charles M. Fleischmann, Patrick J. Pagni and R. Brady Williamson in conjunction with their investigation of backdrafts.

The Lagrangian representation of the fuel elements will allow the incorporation of realistic sub-grid combustion phenomena using models of the type described in Ref. [3]. Work is underway to remove the Boussinesq approximation and to incorporate radiative heat transfer into the model.
References


The interchange between fresh air (white) and heated gases (black) through a window. The Reynolds number is 40,000. The grid size is $1025 \times 256$. No-slip, adiabatic boundary conditions have been imposed.
Fires can produce more products of pyrolysis than the locally available oxygen can consume. If the fire compartment is well-ventilated, these excess pyrolyzates fuel long flames which extend out the compartment opening rapidly spreading the flame. If the compartment is closed, the excess pyrolyzates accumulate, ready to burn when a vent is opened, e.g., by a window breaking due to the fire-induced thermal stress or by a firefighter entering the compartment. The ensuing deflagration moving through the compartment consuming the accumulated excess pyrolyzates after venting is called a backdraft.

Consider a fire in a closed compartment where the only ventilation is provided by leakage. A hot layer composed largely of combustion products descends around the fire causing some pyrolysis products to remain unburned. These products accumulate forming a deep, fuel-rich layer. Suddenly a vent is opened. The hot, vitiated atmosphere within the compartment flows out of the upper portion of the vent. Simultaneously, cold, fresh air flows into the lower portion of the vent. A mixed layer forms due to the instabilities at the shear interface between the outflow and the inflow. Much of this mixed layer is within the flammable range and is ignited when it reaches a flame or glowing ember. After ignition, a rapidly propagating turbulent flame propagates back through the mixed layer. The resulting turbulent deflagration within the compartment drives the remaining accumulated excess pyrolyzates out the vent to be burned outside the compartment.

The gravity current which enters the compartment is an important parameter in understanding backdrafts. The gravity current is responsible for the time delay between the compartment opening and the ignition of the deflagration. This time delay causes fire fighters to become trapped when a backdraft occurs. Salt water experiments were conducted to quantify the height and travel time of the gravity currents. Four different opening configurations over a range of density differences, 0.01 to 0.10, were used. Over this density range the Froude Number, \( V^* = \frac{U}{(g'h)^{1/2}} \), where \( g' \) is \( g \) multiplied by the density difference, was constant. The value of the constant \( V^* \) varied with the geometry of the opening. Figure 1 shows the results for the center 1/3 horizontal strip opening. \( V^* \) was nearly constant at 0.32. Other geometries yielded \( V^* \) ranging from 0.25 to 0.50. Ignition delay times from the 1/2 scale backdraft experiments have compared well with the constant Froude Number results from these experiments.

The average height of the gravity current was also found to be a constant, independent of the density difference,
Half scale backdraft experiments have been conducted to better quantify backdrafts. Measurements included species concentration histories of the upper layer to better characterize the conditions prior to opening the 'compartment. Temperature and pressure measurements were used to estimate the mass flow into and out of the compartment prior to the deflagration. Using these results and an estimated flame speed, a global energy balance can be performed on the compartment to calculate the pressure and temperature of the backdraft. A comparison of the predicted results with the experimental results will be presented.

![Graph](image)

Figure 1 - Shows constant value for $V^*$ versus density difference for the center 1/3 horizontal opening

References


WHAT HE KNOW AND DON'T KNOW ABOUT CEILING JETS
Howard H. Emmons

Abstract

The movement of fire gases along a ceiling from a fire plume or open fire room door appears very simple: just like water in a river. From this we know that the flow may be tranquil or shooting. For the leading edge of an advancing ceiling jet, the speed is close to critical i.e., Richardson Number \( \approx 1 \). When this spreading jet reaches the end of a corridor, the Richardson Number remains one if it is open or if the jet hits a closed end and returns as a hydraulic jump. In any case the corridor end will determine the near source flow which may be shooting flow with a hydraulic jump or may flood the source and cause a "sudden enlargement" flow. All this would just like for water except that the density ratio for the ceiling jet is about 4 to 1 while for water it is 1000 to 1. Thus all of the above phenomena is modified for the ceiling jet by a relatively large entrainment. Hater and air separate after mixing while hot air and cold air do not.

There is a need for extensive knowledge of the values of friction factors and entrainment coefficients which will require many correlatable results or a large program of 3D field model calculations. This is an important use of field models not yet exploited because of cost.

The most critical effect for the ceiling jet, however, is the heat transfer. Experimental studies of this effect cannot be carried out by salt solution tests. Liquids can exchange considerable heat with very small changes in density. The fire ceiling jet looses buoyancy as it cools. It often cools so much that it fall from the ceiling and returns below into the fire. During experiments with cooling, the ceiling jet velocity decreases and the jet becomes deeper for some distance then continues further without change. The simple theory does not level off. In particular the layer can't speed up to critical at the transient front nor at an open end.

To satisfy boundary conditions at the source or corridor end, it is necessary to use the momentum equation perpendicular to the ceiling in order to modify the ceiling jet pressure from the simple hydrostatic. Then this is done, the controlling equation becomes third order, thus the three required boundary conditions can be met. Figure 1 shows the range of solutions that exist. Most of the solutions contain standing waves. One solution contains a constant depth section. For the simplest case \( f = H = E = \theta \), the equation can be integrated twice and then reduced to an integral. This integral shows that for any ceiling jet depth and surface slope, there is one initial surface Curvature which results in a ceiling jet of constant depth in a long corridor. Curiously, all ceiling jets can have a long constant depth section in shooting flow. Such constant depths in tranquil flow do not exist. Figure 2 shows the constant depth \( y_1 \) to be reached by a ceiling jet from a source of depth \( y_1 \) and Surface slope \( \theta_1 \). Figure 2 also shows the required source surface curvatures. A tranquil source has a unique shooting flow constant depth ceiling jet while a shooting flow source has three possible constant depth ceiling jets, two in shooting flow and one in tranquil flow. However only the shooting flows could reach a Richardson Number \( \approx 1 \). The effect of friction and heat transfer have not yet been studied. However the theoretical results already obtained are in need of experimental verification by long corridor (perhaps model scale) tests. It is hoped that study of the effect of heat transfer will answer the question of how real conditions are satisfied.
A field model has many advantages compared with zone models, but there has been a lack of experimental data from large-scale experiments with which to verify the calculations. Therefore, large-scale experiments were carried out at the Swedish National Testing and Research Institute in a large test room, both with and without fire vents. The test room measured $7.5 \times 15$ m and $6$ m in height. The configuration of the test room and the location of the fire source and the fire vents were assumed to cover about 60% of the possible combinations of the real cases for industrial buildings in Scandinavia. The fire source was located at the centre of the test room and the position of the fire vent at the ceiling was varied. The fire vent in the ceiling had the dimension $1 \times 2$ m. Tests were also performed to provide a basis for comparison between experiments and zone models and field model calculations. Tests were also performed to investigate the influence of water spray on the outflow of hot gases through a fire vent. A number of tests were also carried out to estimate the influence of beams in the ceiling on the local velocity and temperature field. The temperature and velocity fields were measured at different locations in the room and in the fire vent opening. Both growing and constant heat release rates were applied. Three different fire growth rates were used, fast, medium and slow.

Different semi-empirical relationships have been developed to predict the gas temperature and velocity at the ceiling for different fire and ceiling conditions. Parameters such as fire size, fire growth rate and room geometry in two dimensions are included in the models. The relationships assume a flat ceiling with no obstructions, and the ceiling jet is assumed to be radially axisymmetric. In many buildings these conditions are not fulfilled as obstructions and vents of different types are mounted in the ceiling. Local effects on the temperature and velocity fields caused by beams, smoke curtains, walls, fire vents or water spray from already activated sprinklers, cannot be satisfactory predicted with semi-empirical models. For these cases field model codes are often required. There are conditions under which the response of a sprinkler are difficult to predict. In the Scandinavian countries most ceilings are not flat as they usually have extensive load bearing systems due to the high snow loads. The problems to be solved are therefore more often three-dimensional than two-dimensional and should therefore be simulated by more advanced methods such as field models. Until now, there has been a lack of full-scale fire tests to verify simulations with field model codes. The experiments presented here provide the basis for the necessary verifications.

The interaction of sprinklers and fire vents has been discussed intensively during the last decade without any satisfactory solution. It may be possible to elucidate this question by using a field model code. However, an investigation of the ability of field models to predict local effects around a fire vent opening and the effects of the sprinkler water spray on the ceiling jet flow must first be undertaken. Therefore, as a first step in this direction, extensive experimental data for the verification of a field model code for such complicated cases are presented here.

The test programme was divided into four major parts:

- Test series with different fire growth rates and fire vent conditions to investigate local effects of the fire vent on the temperature and velocity field.

- Investigate the influence of water spray, discharged from a conventional sprinkler, on the ceiling jet.

- Investigate the effects of water spray on the outflow of hot gases through a fire vent by varying the location of the sprinkler.
-investigate the influence of beam constructions located in the ceiling on the velocity and temperature fields.

The fire vents applied clearly affect the temperature and velocity field in the test room. The temperature reductions varied depending on distance from the fire vent. The observed effects on the temperature and velocity fields are probably out of range for zone models, but possible to predict with field models.

Experiments with sprinklers mounted at the ceiling demonstrated the effects of when the fire vent was located above the fire source. With the fire vent mounted eccentric to the fire source it was not clear if the fire vent had any impact on sprinklers located upstream from the fire vent. From temperature and velocity data it could be observed that the fire vent clearly reduced the temperature and velocity field downstream from the fire vent. Measurements of total heat release rate through the fire vent show that the efficiency of the fire vent varied depending on the location of the fire source in relation to the fire vent.

The impact of sprinkler spray on the ceiling jet was investigated by applying a conventional sprinkler mounted in a pendant position at the centre of the test room. The fire source was moved to one end of the test room and the terminal wall was attached to the ceiling to force the hot gases in one direction. As could be expected, the water flow rate influenced the cooling of the ceiling jet. At centreline instrument stations downstream from the sprinkler spray, the temperature reduction depended on the water flow rate. The temperatures upstream of the sprinkler spray tended to increase in some cases and decrease in others. Some kind of three-dimensional blocking effects are probably the reason for this but the results were not very conclusive. The velocities of the ceiling jet were slightly reduced downstream from the sprinkler spray.

The results show that if the sprinkler is located upstream of the fire vent, it will have a significant impact on the gas flow. The experiments also demonstrated the influence of the sprinkler distance from the fire vent and the water flow rate on the gas flow. With the sprinkler located downstream from the fire vent the influence of the gas flow was negligible. It is clear from these results that the location of the sprinkler affects the gas flow through the fire vent. The impact on the gas flow is greatest when the sprinkler is very close to the fire vent and decreases with the distance.

A beam construction was mounted in the ceiling to investigate the impact on velocity and temperature fields. Two 0.5 m high beams were mounted in the ceiling and the fire source was placed between the beams at floor level. The results show that the beam construction influenced the temperature and velocity field considerably. In the space between the two beams the increase in both temperature and velocity would definitely influence the sprinkler operation. In the spaces between the walls and the beams the velocity and temperature were reduced and the velocity was influenced more than the temperature. The thickness of the ceiling jet was not significantly affected.

References


A review is given of entrainment measurements made in the flaming part of fire plumes. Although the data studied emphasize the work of a group at Caltech (see Cetegen et al. 1982), Toner 1989, and Morehart 1991), measurements of P. H. Thomas et al. (1963), McCaffrey (1979), Beyler (1982), and Delichatsios and Orloff (1984) are also considered. Correlations developed by Cetegen et al. (1982), Delichatsios and Orloff (1984), and Heskestad (1986) are described and a new correlation is presented. The study is limited to flames with heights in the range 1 to 6 burner diameters with $Q^*$ values between 0.1 and 5.

In the early work of P. H. Thomas et al. (1963), fire plumes generated by gas burners and wood cribs were placed beneath a hood like ceiling with short vertical side walls. The vitiated gas was allowed to escape through a vent in the ceiling and from under the sides of the containing walls, and the mass flux in the plume at the interface between hot and ambient gas was estimated from simple formulae used to model the flow through these two openings.

In the measurements made by the Caltech group, a hood, open on the bottom side, was placed over the fire, like a candle snuffer. The hot products of combustion are collected within the hood and escape from the hood along the bottom edges of the sides. A laminar, sharply defined interface develops between the products of combustion contained within the hood and the ambient atmosphere. This interface is penetrated only by the gas in the fire plume and the assumption is made that the only oxygen atoms which enter the hood comes from the fuel or from air which is entrained into the fire plume from the region below this interface. Measurement of the specie concentrations present within the hood then allow the fuel-air ratio of the gas supplying the hood to be determined and given a measurement of the fuel flow rate, the entrainment rate can then be determined.

Both techniques described above measure the mass flux carried into the upper hot-gas layer which is the quantity required for fire models; whether or not the presence of the interface substantially disturbs the entrainment process in the plume has not been determined.

Data collected by the Caltech group and Beyler, who used a similar technique but with a different type of hood, showed unambiguously that for elevations below the top of the flame, the mass flux in the fire plume was linearly proportional to the burner diameter and the distance above the burner, and was substantially independent of the fuel flow rate.

Beyler’s results were qualitatively similar but differed from the results of the Caltech group by about a factor of two when data for similar fuels, heat release rates and fuels are compared.
The results of Thomas et al. also showed that the plume mass flux scales with the burner diameter, (they use the perimeter of their cribs as the scaling parameter), and was independent of the heat release rate in the fire. However, they scaled their dependence on the height of the interface with $Z^{3/2}$ rather than the linear dependence found by Beyler and the Caltech group. The extra $Z^{1/2}$ combined with the acceleration of gravity gives a characteristic velocity which can be used to make the correlation dimensionally correct. Unfortunately, data from the hood experiments show that the mass flux clearly depends on $Z$ not $Z^{1.5}$.

The best fit to the Caltech entrainment data shows that the entrained mass flux at an elevation $Z$ above a burner with diameter $D$ is given by $C_e Z D$. When $Q^*$ is in the range between 0.1 and 6, the value for $C_e$ is a constant equal to 0.62 kg/sm$^2$ and this proportionality constant decreases with decreasing $Q^*$ below 0.1 and increases linearly with increasing $Q^*$ values above 6. A scaling parameter which has the correct dimensions and which depends linearly on the diameter and height, and is independent of the heat release rate has not been found. The parameter proposed by Thomas et al. is intuitively satisfactory but does not fit the Caltech or Beyler data.

REFERENCES


SCALAR MIXING PROPERTIES
OF BUOYANT TURBULENT PLUMES

by

Z. Dai, L.-K. Tseng and G. M. Faeth
Department of Aerospace Engineering
The University of Michigan
Ann Arbor, Michigan 48109-2140

The scalar mixing properties of round buoyant turbulent plumes were measured in still air. This is an important fundamental problem that has attracted significant attention since the classical work of Rouse et al.; see List, George et al. and references cited therein for discussions of research during the intervening years. However, recent work in this laboratory has highlighted the need for additional information concerning the turbulence properties of scalar quantities within buoyant plumes, in order to address turbulence/radiation interactions in fire environments, see Kounalakis et al. and references cited therein. In particular, the response of flame radiation to turbulent fluctuations is particularly sensitive to the moments, probability density functions, and spatial and temporal correlations of mixture fraction fluctuations. While the classical work of Pitts and Kashiwagi provides information of this type for nonbuoyant turbulent jets, surprisingly, comparable information is not available for buoyant turbulent plumes. Thus, the objective of the present investigation was to complete measurements of mixture fraction statistics in round buoyant turbulent plumes, and to interpret and correlate the results in order to facilitate treatment of turbulence/radiation interactions in flames.

The experiments employed techniques developed by Lai and coworkers. This involved using mixtures of carbon dioxide and air, and observing the resulting negatively-buoyant plumes. Initial conditions consisted of fully-developed turbulent pipe flow, with mean velocities adjusted so that the flow corresponded to the asymptotic Froude number of round turbulent plumes in order to enhance rates of development toward fully-developed turbulent plume conditions. Effects of room disturbances were controlled by observing the flow in a screened enclosure surrounded by a plastic enclosure, all within a windowless room, with the plume exhausted using a distributed collection system located in the corners of the plastic enclosure. The mixture fraction measurements were made using laser-induced iodine fluorescence (LIF) based on the 514.5 nm line of a cw argon-ion laser (with roughly 1500 mW of optical power). The laser beam passed horizontally across the flow and was not focussed, so that two traversible detectors could be used to measure the radial spatial correlations of mixture fraction fluctuations—a rarely measured property of turbulent mixing that is crucial for understanding turbulence/radiation interactions.

Various positions in the flow could be studied by traversing the screened enclosure in the horizontal (radial) direction, and the plume generator in the streamwise direction within the screened enclosure.

We expected the measurements of mean mixture fractions in the plumes to serve primarily as a check of the operation of the present apparatus, however, this was not entirely the case and these measurements were pursued more thoroughly than planned. In particular, while existing measurements are seemingly in the fully-developed portions of the plumes, based on a Monin number criterion, they are actually at relatively low (x-

---

Research supported by the Building and Fire Research Laboratory of the National Institute of Standards and Technology, Grant No. 60NANB1D1175.
$x_0/d$, ca. 20-30, where $x$ and $x_0$ are the streamwise distance and the position of the virtual origin and $d$ is the plume generator diameter.\textsuperscript{1-3} Thus, past flows generally are not fully-developed and present measurements indicate somewhat narrower plume widths in fully-developed portions of the flow, for $(x-x_0)/d = O(100)$. Within this regime, however, the classical similarity behavior proposed by Rouse et al.\textsuperscript{1} is observed and entrainment rates agree with results in the literature within experimental uncertainties.

Radial profiles of mixture fraction fluctuations confirmed our findings concerning the development of mean property distributions with streamwise distance. Near the plume generator, mixture fraction fluctuations are relatively low and exhibit a dip to a local minimum near the axis. With increasing distance, however, the dip disappears and fluctuation intensities near the axis stabilize at roughly 40%, which is higher than existing results in the literature.\textsuperscript{2} Probability density functions of mixture fraction are represented reasonably well by the clipped-Gaussian functions, while the behavior of the popular beta function distribution is somewhat less satisfactory. Temporal power spectra exhibited a limited inertial range because present measurements are still at modest Reynolds numbers: current work is seeking to remove this limitation. Additionally, the results show that effects of signal noise on present measurements are small. Finally, present two-point measurements of radial spatial correlations are in qualitative agreement with earlier results for forced-flow conditions in flames.\textsuperscript{4} Work is currently in progress to develop corresponding information for streamwise spatial correlations and temporal and spatial integral scales.

References

MOMENTUM–DIFFUSION CONTROLLED LAMINAR DIFFUSION FLAMES

H. Ban, S. Venkatesh and K. Saito
Combustion and Fire Research Laboratory
Dept. of Mechanical Engineering
University of Kentucky
Lexington, KY 40506

ABSTRACT: Small laminar diffusion flames (flame height \( \approx 2-3 \) mm) established by issuing a fuel jet into a quiescent medium are investigated. It was found that for these flames buoyancy effects disappeared as the flame size decreased (Fr >> 1), and diffusive transport of the fuel was comparable to the momentum transport of the fuel. The effect of buoyancy on these flames was determined by examining the flame shape for horizontally oriented burners. A phenomenological model was developed (based on experimentally determined flame shapes) to compare diffusion and momentum transport effects. Finally, the flame shapes were theoretically determined by solving the conservation equations using similarity methods. It was seen that when the axial diffusion (viscosity) terms are included in the conservation equations, the calculated flame shape is in better agreement (as compared to without the axial diffusion term) with the experimentally measured flame shape.

Studies on laminar hydrocarbon diffusion flames are always of interest because of their fundamental nature. Beginning with Faraday’s pioneering work - "Chemical History of a Candle," in 1861, many studies have been conducted on this subject, and that there is still a need to study them was further established through recent observations by Ban, et al. (1992), where small laminar diffusion flames (with flame heights of a few millimeters) were studied, by issuing a hydrocarbon fuel jet into a quiescent atmosphere. They found that the flame shape of these small flames is spherical and different from the more commonly observed candle-like laminar diffusion flames, and for sufficiently small flame sizes there were no buoyancy effects. These observations led to the current investigations on the physical characteristics of the small flames, since we were unable to find published literature on this class of flames. Experiments were conducted to collect data on the flame height and flame shape as a function of flow rate for three different fuels. It was found that the small flames have interesting characteristics that allow the phenomenological evaluation of the effects of molecular diffusion, momentum and buoyancy.

Based on past studies, laminar jet diffusion flames can be classified into two main types: The Burke–Schumann (1928) flame controlled by diffusion, and the Roper flame (1977) controlled by buoyancy. The small flames studied here have the following characteristics: Fr >> 1 and \( u_e \approx u_D \), where \( u_e \) is the fuel exit velocity at the burner port, \( u_D \) is the molecular diffusion velocity,
\[ Fr = \frac{u_e^2}{gd} \] and \( d \) is the burner port diameter. A scale analysis reveals that for \( Fr \geq 1, u_e \approx 0.1 \text{ m/s} \) and \( d \approx 1 \text{ mm} \) the buoyancy effects can be neglected (Williams, 1985). In addition, for small flames \( u_e \approx v_D \) implying they are momentum-diffusion controlled.

**EXPERIMENTAL:** The flames were established on circular-port stainless-steel burners with inner diameters 0.15, 0.25 and 0.40 mm; the outer diameters were twice the inner diameters. Research grade ethane, ethylene and acetylene were used as the fuels. The flow rates of the fuels were calibrated using a specially designed soap-bubble flow meter capable of measuring low flow rates of order, 0.01 cc/s. The fuel was issued into the quiescent ambient at atmospheric conditions. The flame shape was recorded by a 35 mm still-camera with a bellows attachment to magnify the image.

**THEORETICAL:** To explain the observed experimental phenomena, a simple theoretical model was developed. The case of a combustible laminar jet issuing into a quiescent medium was theoretically described by Spalding (1979). Spalding's analysis uses the similarity procedure described by Schlichting to develop closed form expressions for the flame height and flame shape for circular jet laminar diffusion flames. Governing equations for the small flames for axisymmetric geometry were derived. With the assumptions of negligible buoyancy, constant pressure, and constant property values, the flame height and flame shape of the small laminar momentum-diffusion controlled flames were solved using similarity analysis for two cases, one is with axial diffusion term and the other is without axial diffusion term. The comparison with experimental data proved that transport by axial diffusion is comparable to the convective transport and cannot be neglected when determining the flame height/shape in such a small flame.

**SUMMARY:** (1) Small laminar 'momentum-diffusion controlled diffusion flames, with Pe numbers \( u_e l_D/D \) less than 5 were investigated. It was observed that these flames tend to achieve a spherical shape due to the comparable effects of momentum and molecular-diffusion.
(2) It was found that the effect of buoyancy in these flames is negligible, by rotating the burner. The Froude numbers of these flames were in the range \( 10^{-1} \text{ to } 10^3 \).
(3) A phenomenological model was developed to understand the flame shape and estimate the momentum/diffusion effects.
(4) The flame height/shape was theoretically determined starting with the governing equations and applying similarity methods. It was seen that when axial diffusion terms are used in the governing equations, the calculated flame shape is in better agreement (as compared without the axial diffusion terms) with the experimentally determined flame shape.
(5) For further details of this work, see the same title paper by H. Ban, S. Venkatesh and K. Saito, at the 1992 ASME Winter Annual Meeting, November 8-13, 1992, Anaheim, CA.
MICROSCALES OF FORCED/BUOYANCY DRIVEN TURBULENT DIFFUSION FLAMES

V. S. ARPACI
Department of Mechanical Engineering and Applied Mechanics
The University of Michigan

R. J. TABACZYNSKI
Scientific Laboratories
Ford Motor Company

ABSTRACT

The recent decades have seen exponential growth of research on algebraic and transport models for prediction of turbulent flows. The potential of these models now appears to be exhausted and any future research along these lines is expected to be mostly in the nature of "utilization" or "elaboration". A novel approach is proposed here to study the fuel consumption in reacting turbulent flows. The approach is based on the development of appropriate microscales and the use of these scales in the prediction of fuel consumption in furnaces and heaters.

The Kolmogorov scale was originally introduced by intuitive arguments with two kinematic concepts, the dissipation of energy per unit mass,

\[ \epsilon \equiv L^2/T^3, \]

and the kinematic viscosity,

\[ \nu \equiv L^2/T. \]

The elimination of time between these readily gives this scale,

\[ \eta = \left( \frac{\nu^3}{\epsilon} \right)^{1/4}. \]

Extension of the Kolmogorov scale to thermal scales by Oboukhov-Corrsin and Batchelor follows similar arguments. However, the microscales appropriate for more complicated turbulent flows (say those for pulsating or reacting flows) cannot be obtained by similar arguments because of the need of combinations involving more than two concepts such as \( \nu \) and \( \epsilon \).

During the past decade, Arpaci and co-workers originated a methodology for construction of microscales appropriate for complex turbulent heat and mass transfer problems and utilized these scales in the correlation of heat transfer in pulse combustor tailpipes, that of mass consumption in pool fires and from forced flames. For example, the Reynolds and phase averaged turbulent flows pulsating with frequency \( \omega \) leads to the Kolmogorov scale,

\[ \eta = \frac{(\nu^3/\epsilon)^{1/4}}{[1 + \omega(\nu/\epsilon)^{1/2}]^{1/2}}. \]
whose two limits for $w \to 0$ and $w \to \infty$ are the usual Kolmogorov and Stokes scales, respectively. Also, the Reynolds averaged, buoyancy driven, steady turbulent flows lead to the thermal Kolmogorov scales,

$$\eta_\theta = \left(1 + \frac{1}{Pr}\right)^{1/4}(\frac{\nu a^2}{B})^{1/4}, \quad Pr \geq 1$$

and

$$\eta_\theta = \left(1 + \frac{1}{Pr}\right)^{1/4}(\frac{a^3}{B})^{1/4}, \quad Pr \leq 1.$$  

Of these scales, the limit of the first one for $Pr \to \infty$ is the Batchelor scale, and that of the second one for $Pr \to 0$ is the Oboukhov-Corrsin scale. Here $B$ denotes the production of buoyant turbulent energy, other notation being conventional. Also, for the Reynolds averaged, forced, steady turbulent flows, a thermal mesomicroscale between the Kolmogorov and Batchelor scales

$$\delta_\theta = \eta^{1/3}(\eta_\theta^B)^{2/3} = (\nu^{5/3}a^{4/3}/\epsilon)^{1/4}, \quad Pr \geq 1$$

is proposed. Here $\eta$ and $\eta_\theta^B$ respectively denote the Kolmogorov and Batchelor scales.

It is shown by extending this methodology to microscales appropriate for reacting turbulent flows how these scale can be employed to predict the fuel consumption in pool fires and forced flames. The results of the proposed research are expected to be useful to present federal and industrial research programs on combustion, flame and fires.

REFERENCES


MULTIPLICITY OF PUFFING FREQUENCIES IN BUOYANT DIFFUSION FLAMES*

by
Baki M. Cetegen and Tarek Ahmed
Mechanical Engineering Department
University of Connecticut, Storrs, CT 06269-3139

Abstract:
Buoyant propane diffusion flames established on 0.05 and 0.10 m. diameter burners were found to exhibit splitting of puffing frequencies with changes in the fuel flow rate or total heat release of the flames. A series of experiments were performed with flames established on porous-bed pool fire burners in quiescent surroundings to explore this phenomenon. Burner surface pressure fluctuations along with the static pressure at a height of 1/2D above the burner surface were measured to determine the signature of puffing as detailed in a recent article [1]. At low fuel flow rates, the puffing frequency more or less scales with the well-known, diameter dependent frequency (f \propto D^{1/2}). As the fuel flow rate is increased, a higher frequency emerges in both the surface and the static pressure frequency spectrum as shown in the figure. At intermediate flow rates (e.g. Q_f = 6.0 kW), both frequencies have comparable strengths. The higher frequency that appears in the spectra is believed to be associated with the buoyant flame instabilities with a frequency of around 10 Hz. These types of flames have been extensively studied by a number of researchers [2,3]. It is also interesting to point out that the frequency of this instability has been experimentally found to be insensitive to burner size, fuel type, etc. At even higher flow rates of fuel (e.g. Q_f = 8.4 kW), this higher frequency begins to dominate while the amplitudes of the lower frequency fluctuations diminish. Multiplicity of flame frequencies were also recently observed by Sui et al. [4] who reported flame frequencies in coflowing propane diffusion flames. The existence of the lower frequency along with the higher frequencies provide some clues as to which modes of instabilities are in play in transitional diffusion flames. The discussion of these new results will center around the various instability modes and the flame parameters which determine the transitions in the flame behavior as a result of the aforementioned instabilities.

References:

(*) This research has been sponsored by National Science Foundation Grant CTS-8909176
Frequency spectra and pressure fluctuation traces of propane flames on a 10 cm. diameter burner.
A THERMAL ELEMENT MODEL FOR COMBUSTION PROCESSES IN FIRES

Ofođike A. Ezekoye and Howard R. Baum
National Institute of Standards and Technology
Gaithersburg, MD 20899

A methodology for incorporating combustion and radiative processes into large eddy simulations of fire driven flows is presented. The mathematical models describing these processes are developed in a local Lagrangian frame of reference moving with the large scale convection induced by the fire. The gaseous fuel is represented by a large number of discrete "thermal elements" which react with their local oxidizing environment and exchange energy radiatively with each other and with their surroundings. Viewed at large scales which characterize the overall geometry of the scenario under investigation, each element is a point source of energy and volume. At a local scale, the thermal element is a small sphere of gasified fuel being consumed by a transient diffusion-controlled flame. A prescribed fraction of the fuel is converted to soot which can emit radiation to its surroundings or absorb radiation from other thermal elements.

The large and small-scale pictures are coupled both hydrodynamically and thermally. The location in space and time of each thermal element requires a particle tracking computation of its Lagrangian coordinates. The local asymptotic expansion, the chemical and net radiative heat sources in each element are fed back into the computational mesh used to calculate the large scale flow. The net radiation absorbed by each element from all others is approximately accounted for using an optically thin model of radiative transport. The overall velocity field induced by the fire is decomposed into an irrotational expansion field and a solenoidal buoyancy induced flow. The thermal elements generate the expansion field and provide the thermal sources for the large scale temperature field. A large eddy simulation of the Navier-Stokes equations (described elsewhere) generates the solenoidal field.

Mathematically, the local thermal element is described by an unsteady, spherically symmetric diffusion-convection equation for the mixture fraction, a transport equation (including thermophoretic diffusion) for the soot which is assumed to be generated at the flame sheet, and an energy equation which accounts for energy release at the flame sheet and distributed radiation from the soot layer. The gas velocity is induced by the thermal expansion using an analytical recipe derived from the conservation of mass. The total time to consume a fuel sphere of radius R is approximately 10 times a characteristic diffusion time \( \tau \), based on R. The flame sheet expands out in search of air until approximately \( 37 \), and then collapses when the fuel diffusion rate can no longer maintain the stoichiometric mixture fraction value. For a 1 cm. propane diffusion flame with \( \tau = 1151 \) seconds, Figure 1 shows the total heat release and radiative loss rates. At early times, the large gradients in the mixture fraction profile imply large fuel diffusion rates into the flame sheet and thus large heat release rates while at later times, the heat release rates drop due to the shallower gradients. Figure 2 shows the cumulative fraction of chemical energy lost through radiative transfer. In Figure 3, contours of the temperature field (solid lines), the stoichiometric mixture fraction value (0.059, dashed line) and a soot volume fraction value (5.0x10^-8, dotted lines) are shown spatially and temporally. The maximum soot volume fraction during the combustion process is approximately 5.0x10^-7.

The system of equations for a single thermal element is solved using the package MOL1D and can be calculated in several seconds of computer time on a Cray Y/MP computer.
Figure 1. Time history of chemical energy release rate and radiative emission for a 1 cm. propane element.

Figure 2. Cumulative fraction of chemical energy lost through radiation.

Figure 3. Contour plot of temperature in Kelvin (solid lines), stoichiometric mixture fraction (dashed line) and soot volume fraction of 0.05 ppm (dotted line).
COMPARISON OF EXPERIMENTAL AND COMPUTED SPECIES CONCENTRATION AND TEMPERATURE PROFILES IN LAMINAR, TWO-DIMENSIONAL CH₄/AIR DIFFUSION FLAMES

Kermit C. Smyth and Thomas S. Norton
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

J. Houston Miller
Department of Chemistry
The George Washington University
Washington, DC 20052

and Mitchell D. Smooke
Department of Mechanical Engineering
Yale University
New Haven, CT 06520

The last decade has witnessed significant advances in our knowledge of both the chemical reactions which occur under high-temperature combustion conditions and the associated fluid mechanical mixing processes. As a consequence, increasing attention is now focussed on improving our understanding of the typically strong coupling between chemical heat release and turbulent mixing, i.e., chemistry-turbulence interactions. One approach for including chemical reactions in computational models of turbulent diffusion flames is to utilize libraries of strained laminar flame calculations which incorporate detailed reaction mechanisms. An alternative avenue is to reduce systematically the number of chemical steps in the basic oxidation mechanisms for combustion to as few as possible (typically three or four), while still providing an adequate description of the major species concentration and temperature profiles.

Few studies have addressed the question of how well flame structure calculations reproduce experimental results. Recent work at Sandia, mostly in turbulent jet diffusion flames of diluted methane, has compared measured concentrations of the OH radical and the major stable species to values computed for strained (counterflow) laminar flames. With respect to developing reduced chemical mechanisms, most of the effort has used a detailed flame structure calculation as the point of reference. Although both strategies are logical, to this point only limited comparisons have been made between experimental measurements and detailed flame structure computations for laminar diffusion flames, particularly for the radical species.

In the last several years we have undertaken an extensive experimental investigation of a laminar, CH₄/air diffusion flame burning at atmospheric pressure, in which measurements have been made of species concentrations, velocity, and temperature. In addition to mass spectrometric detection of stable molecules, radical concentration profiles have been obtained for OH, H atom, O atom, CH, and CH₃. Beyond these results, profile data for radicals in laminar hydrocarbon diffusion flames are scarce. Melvin and Moss (1975) have reported an OH profile in a N₂-diluted CH₄/air flame, Bastin et al. (1987) have used molecular beam mass spectrometry to sample numerous radicals in a 30-torr C₂H₂/O₂/Ar flame, and Barlow and Collignon (1991) have obtained OH profiles in both counterflow and co-flow air-diluted methane flames.

The detailed experimental profile measurements which are now available in the undiluted CH₄/air diffusion flame enable a much more complete comparison to be made with flame structure calculations than has been possible before. Our initial focus has been on the radical species H, O, OH, CH, and CH₃, since minor species concentrations generally provide more critical and demanding tests of mechanism and model predictions.
than do the major species. However, in our analysis it has become evident that the major stable species also need attention.

Some initial comparisons between experimental measurements and flame structure calculations have been carried out for two-dimensional axisymmetric CH₄/air diffusion flames by Smooke et al. (1989, 1990). The agreement is generally good for the major stable species and provides confidence in the chemical mechanism used in the flame structure calculations. The present effort extends the comparisons between experimental and flame structure predictions in two-dimensional CH₄/air diffusion flames with particular emphasis on the radical species. Since our experimental measurements have been made on a rectangular Wolfhard-Parker burner geometry (flame height = 33 cm), in contrast to the axisymmetric geometry of the flame structure computation (flame height = 3.8 cm), it is also necessary to examine the results using geometry-independent coordinates, such as the local mixture fraction and the scalar dissipation rate. These parameters are of interest for describing the chemical composition and the strain field in laminar flamelets approaches. In order to address the question of which library of laminar flamelets is the best to use for turbulent flame computations, comparisons will also need to be made between counterflow and co-flow geometries. The present study is a first step in assessing how well comparisons can be made for diffusion flames in geometry-independent coordinates.

Our comparisons of temperature and species concentrations have been made by basing the analysis on the local flame conditions, specifically the mixture fraction and the scalar dissipation rate. The best method for establishing a basis of comparison for the scalar dissipation rate involves using a least-squares procedure over a range of mixture fraction, rather than matching at a specific point which corresponds to the stoichiometric surface or the temperature maximum. Good agreement between experimental measurements and calculated concentrations is found for the major radical species OH⁻, H atom, and O atom, while for the hydrocarbon radicals CH⁻ and CH₃⁻, some features of the profiles agree well but significant areas of disagreement exist. Overall, the calculated positions of the peak radical concentrations are slightly more compressed in terms of the mixture fraction compared to the experimental results. The predicted concentration profiles for the major species certainly agree with the experimental measurements as well as found in investigations on premixed flames, for example in acetylene combustion (Westmoreland et al., 1986). The computed results are somewhat more sensitive to the scalar dissipation rate than are the experimental profiles.

The most serious areas of disagreement concern the comparison of the temperature profiles and the much larger degree of O₂ penetration in the computed flame structure. The influence of the latter, particularly in rich flame regions, is likely to be small and can be further investigated by carrying out one-dimensional counterflow calculations with and without a small amount of O₂ added to the fuel stream and then comparing species profiles. Given the importance of the temperature field in determining the chemical structure of a diffusion flame, the disagreement between the computed and the experimental temperature profiles certainly deserves further study before one can be satisfied with the generally excellent agreement for the individual species profiles.

1 Submitted to Combustion Science and Technology.

2 NRC-NIST Postdoctoral Research Associate (1990-1991); current address: Morgantown Energy Technology Center, Morgantown, West Virginia 26505.
Numerical Simulation of Thermal Plumes

Soonil Nam, Robert G. Bill, Jr.

Factory Mutual Research Corporation
Norwood, Massachusetts 02062, USA

Thermal plumes generated by pool fires and heptane spray fires were simulated numerically with the aid of the computer code PHOENICS. The flow fields associated with the various heat release rates were simulated and compared with experimental results. The experimental results were obtained from five different fires: methanol pool fire (denoted to Test 1), hydrocarbon fluid pool fire (Test 6), heptane pool fire (Test 8), Heptane Spray Fire 1 (HS1) and Heptane Spray Fire 2 (HS2). Modifications were made in the standard $k-\varepsilon$ model used in the PHOENICS code in order to improve the accuracy of that model in the simulation of turbulent thermal plumes. The modification includes an additional term related to the buoyancy generated turbulence kinetic energy, changing constants responsible for the turbulent viscosity and the effective Prandtl number in the energy equation. The results obtained with the modified code were in excellent agreement with experimental results and significantly improved the results obtained with the standard $k-\varepsilon$ model. The standards of the comparison were the well established plume relations based on axial velocity, $v_{z0}$, the excess temperature above ambient at the centerline, $\Delta T_0$, and the plume half-width based on the centerline temperature, $b \Delta T$. Thermal plumes generated with the modified model give excellent agreements with the plume coefficients reported in Reference 1. Figure 1 shows the comparison based on one of the coefficients $C_{\Delta T_0}$ which is a constant relating $\Delta T_0$ with the heat release rate and elevation.

![Figure 1](image)

Figures 2 and 3 show the comparison between the experimental and numerical results for Heptane Spray Fire 2. Air entrainment rates were computed and compared with the formula developed by Heskestad, a comparison is given in Figure 4. The simulation was extended to calculate the ceiling jet flow under an unconfined flat ceiling with 3 different ceiling heights, H, with two different fire sizes (HS1 and HS2).
The results are shown in Figure 5 and are compared with other experimental data available. Here $AT_0$ denotes the maximum excess temperature in the ceiling jet. $\Delta T_0$ is the excess temperature at the centerline of the undeflected plume at the level of ceiling height $H$, $r$ is the radius along the ceiling, $b$ is the half-width of the undeflected plume at the level of $H$. The simulations with the present model significantly improve the agreements with the experimental results of buoyant thermal plumes.

REFERENCES

THE TURBULENCE STRUCTURE OF MEDIUM-SCALE POOL FIRES

E.J. Weckman and A.B Strong
Department of Mechanical Engineering
University of Waterloo, Waterloo, Ontario, Canada, N2L 3G1

Introduction

Explanations of mixing and entrainment in pool fires, and therefore full mathematical models of fire behaviour, are only in their formative stages. The reasons are twofold. First, only a very limited number of detailed measurements of local velocities, temperatures and species concentration have been reported due to difficulties in making measurements near the base of the fire. Secondly, the data that are available typically consist of measurements of only one or two parameters, i.e., mean temperature and concentration [1], or mean temperature and velocity [2,3,4,5] over restricted regions in the fire flow field. Determinations of the correlations between parameters, the key to understanding mixing and entrainment, have been limited to some autocorrelations [2] and only a few Reynolds stress and axial velocity and temperature correlations [2,4,5].

In this study, the flaming regions of 0.31 m diameter methanol pool fires were measured. A full set of simultaneous velocity and temperature measurements were made in and around the continuous and intermittent flame zones of the fires. Mean and rms values were calculated for each parameter from time series of data. In addition, time correlations, \(u'v'(\tau), u'T'(\tau)\) and \(v'T'(\tau)\), turbulence length and time scales, turbulence Reynolds number, \(R_e_t\), and flux Richardson number, \(R_i_f\), were estimated throughout the measured flow domain. The combined results were used to examine the mechanisms of production of turbulent energy for various points in the fire.

The Experiment

The fires were established in the existing University of Waterloo pool fire test facility [4]. Axial and radial components of velocity were measured using two component LDA operated in the forward scatter configuration, while temperature was measured with a 50 \(\mu\)m diameter bare wire Pt/Pt-10%Rh thermocouple. The data acquisition system allowed simultaneous sampling of four channels of velocity and temperature data. Time series of data consisting of 40,000 measurements were analysed to provide mean and rms values, correlations, length and time scales for each parameter.

Results

The mean temperature and velocity fields substantiate existing descriptions of the fire flow field [5,7,8]. The development of the mean velocity and temperature fields closely parallel one another since buoyancy resulting from heat release in the fire is the major source of vertical momentum in the flow. The velocity profiles are narrower than the temperature profiles, so that regions of highest radial gradient of axial velocity are closer to the fire centreline than the highest radial temperature gradients. In this respect, the development of the mean flow field in the pool fire parallels that in a burning gasjet, suggesting strong similarities in the overall mixing and entrainment processes.

In order to discuss the role of buoyancy in development of the flow, as well as its link to turbulence properties, contributions to turbulent energy production due to the mean flow were considered as distinct from those due to buoyancy. Measurements of the most significant terms in the kinetic energy balance indicate that, in the fire, the dominant path for buoyant production of turbulent kinetic energy is the indirect route from \(\beta g T^2\) production-of the \(u'T'\) scalar flux, feeding buoyant production of kinetic energy through \(\beta g u'T'\).

The magnitude of \(\beta g T^2\) is fairly uniform throughout the fire flow field, except very near the pool rim. There, enhanced mixing due to entrained ambient air feeds the flapping flame fronts which cause large relative temperature intensities. These intensities directly feed the generation of scalar flux \(u'T'\) and buoyant generation of turbulent kinetic energy. Contrary to the uniform axial production of energy through temperature intensity, the net buoyant contribution to turbulent kinetic energy rises rapidly with height above the fuel surface and falls off in the radial direction (Fig. 1). Evidently, the mean flow production terms, through the mean flow gradients, play a significant role in the balance of scalar flux, \(u'T'\).

The relative importance of buoyancy and shear production of turbulent kinetic energy was also examined. The contribution to shear production of turbulent kinetic energy (Fig. 2) via \(-u'v'/\partial U/\partial r\) is dominant through most of the flow, with \(-u'v'/\partial U/\partial r\) contributing significantly (up to 30%) only in the outer regions of the fire well above the fire base. The topology of the shear and buoyancy production terms are similar, with the highest...
levels of shear production corresponding to regions of highest mean velocity gradient. The flux Richardson number, $R_i = -G_{buoyancy}/P_{shear}$, is approximately $-1.0$ throughout the fire flow field, much larger than would be expected in a typical buoyant jet and approaching values seen in large-scale atmospheric flows.

A turbulence Reynolds number, $Re_t = \sqrt{\kappa \lambda u'/\nu}$, was calculated based on temperature corrected viscosity, and calculated Taylor microscales for $u'$. Levels of turbulent kinetic energy increase with height above the fuel but decrease toward the edges of the fire, while the Taylor microscale for length increases in value in the radial direction. The combination of these effects lead to fairly uniform values of $Re_t$ radially, but increasing with height above the fuel. Choosing a critical value of $Re_t$ of 100 for fully turbulent flow, the data indicate that the fire is turbulent over most of the domain and suggest a very limited region (less than 0.1 m above the fuel surface) where laminar diffusion could be considered an appropriate mechanism controlling combustion.

**Conclusions**

The velocity and temperature data presented here substantiate many of the observations made in previous studies of medium-scale pool fires.

New information is provided through calculations of velocity and temperature turbulence properties. A preliminary evaluation of these measurements shows that high values of some properties initially occur in regions of flapping flame fronts. The profiles of all turbulence properties evolve in a manner consistent with those in a buoyant shear flow but for the relatively higher negative values of flux Richardson number which suggest aspects of large scale environmental flows. Clearly, the flow is turbulent through most of the fire.

The detailed data presented contribute to a complete mapping of the coupled, fluctuating velocity and temperature fields in a liquid pool fire. They provide an entirely consistent description of fire flow field development and contribute to the data base required to improve and validate existing explanations of fire development.

**References**


![Fig. 1: Buoyancy production of turbulent kinetic energy due to scalar flux $u'T'$](image1.png)

![Fig. 2: Total contribution of shear production to the balance of turbulent kinetic energy due to $(-\frac{\partial u'}{\partial r} \overrightarrow{\alpha u} - \frac{\partial u'}{\partial z} \overrightarrow{\alpha v})$](image2.png)
RADIATIVE PROPERTIES OF AN UNSTEADY JET DIFFUSION FLAME

Carolyn R. Kaplan, Seung W. Baek*, and Elaine S. Oran**
Navy Technology Center for Safety and Survivability
Naval Research Laboratory
Washington, D.C. 20375

INTRODUCTION

Numerical simulations of an axisymmetric buoyant ethylene jet diffusion flame have been carried out to study the effects of radiation transport on the development, structure and dynamics of the flame. The numerical model is based on one originally developed by Elzey et al. [1] for hydrogen jet diffusion flames. Now, it has been modified to examine hydrocarbon flames, so that it includes new models for chemical reactions and energy release, soot formation and radiation transport. In this paper, we simulate a low-velocity laminar ethylene diffusion flame for which experimental data are available in the literature [2]. These simulations serve as a benchmark for validation of these new algorithms.

NUMERICAL METHOD AND MODEL FORMULATION

The numerical model solves the time-dependent equations for conservation of mass density, momentum, energy, individual species number densities, soot number density and soot volume fraction. These equations include terms for convection, thermal conduction, molecular diffusion, viscosity, gravity, chemical reaction and energy release, soot formation and thermal radiation. The equations are then rewritten in terms of finite-volume approximations on an Eulerian mesh and solved numerically for specified boundary and initial conditions. The model consists of separate algorithms for each of the individual processes, which are then coupled together by the method of timestep splitting [3]. A thorough description of the algorithms for convection, thermal conduction, molecular diffusion, viscosity and the coupling of the individual processes have been previously described in detail [1]. The chemical reaction and energy release process for ethylene oxidation is modeled phenomenologically using a finite-rate, quasi-global Arrhenius expression from Westbrook and Dryer [4]. The evolution of soot number density and volume fraction is represented by two coupled ordinary differential equations derived by Moss et al. [5] which includes terms for soot nucleation, surface growth and coagulation. The radiative heat flux is found from a solution of the radiative transfer equation using the Discrete Ordinates Method [6]. The radiation transport model spans twelve directions and includes radiative effects from soot, CO₂ and H₂O. Scattering is neglected since it is negligible in comparison to absorption.

We simulate a case in which undiluted ethylene fuel flows at 5.3 cm/s through a 1 cm diameter burner into a coflowing air stream. The computational domain covers a region of 10 cm x 15 cm for a grid consisting of 64x88 cells. The left hand boundary is an axis of symmetry, while the right hand boundary is a free-slip wall. The bottom boundary represents an inflow condition, while the top is an outflow boundary where the pressure is adjusted to atmospheric conditions. Boundary conditions for the radiation transport model include diffusely emitting-reflecting surface conditions at the inflow, outflow and right hand boundaries. At the axis of symmetry, the specularly reflecting boundary condition is used. Timesteps are on the order of 10 μs.

*Department of Aerospace Engineering, Korea Advanced Institute of Science and Technology, 373-1 Gusung-dong, Yusung-ku, Daejon, Korea
**Laboratory for Computational Physics and Fluid Dynamics, Naval Research Laboratory, Washington, D.C. 20375

RESULTS

Figure 1 shows instantaneous contours of mixture fraction, soot volume fraction, temperature, radiative intensity and the radiative heat flux vectors at 40000 timesteps. Mixture fraction is defined as the mass fraction of fuel elements found in the mixture at any location in the flow. The maximum flame temperature, 2100 °K, is located 4 cm from the base of the flame. The maximum soot concentration is 16x10⁻⁶, and is located within the region where the mixture fraction ranges from approximately 0.05 to 0.1. The maximum mole fractions of H₂O and CO₂ (not shown) are 0.11 and 0.12, respectively, and are located in the high-temperature region. The temperature contours show the low-frequency buoyancy-driven outer structures which are associated with flame flicker.

The radiative intensity is greatest near the sooting region, with a maximum value of 1000 kW/m², and then decreases with distance from that region. The contours show the strong attenuation of radiative intensity in the heavily sooting region. The length of each of the radiative heat flux vectors is proportional to the magnitude of the vector. The vectors with the largest magnitude are located near the sooting region and decrease with distance from that region. Although the radiative flux vectors point predominately in the radial direction, there is a strong axial component which follows the curvature of the sooting region. This indicates that the axial component of radiation transfer cannot necessarily be neglected, which is common practice in many radiation transport models.

REFERENCES


Fig. 1. Instantaneous contours at timestep 40000 (0.4 s)
ADDITIVE EFFECTS ON SOOT FORMATION IN LABORATORY POOL FIRES

E.J. Weckman and C. Salvador
Department of Mechanical Engineering
University of Waterloo, Waterloo, Ontario, Canada, N2L 3G1

Introduction

The effects of additives on soot formation and burnout in flames have been studied for many years [1-6]. It has been determined that inorganic alkali and alkaline earth salts [1-3], as well as organo-iron compounds [4-6] can have marked effects on the overall soot production in both premixed and diffusion laboratory-scale flames. In the case of organo-iron compounds, it appears that soot formation initially is enhanced, with large numbers of small soot particles being produced [4]. Following this, depending on the total residence time of the soot in the flame, the additive appears to have either increased [4] or decreased [5] the total amount of soot produced in the flame. Recently, it has been demonstrated that organo-iron compounds decrease total soot emission from both small-scale kerosene and crude oil fires [5,6] and larger-scale crude oil fires [7]. Chemical mechanisms have been proposed to explain the effects of these additives on the soot formation mechanisms in these open fires, but no quantitative measurements were made in the fire flow field. Therefore, further study is required in order to verify details of the additive/fire interactions in open diffusion flames or to predict the effects of an additive on soot production in larger-scale fires.

In this study, the flaming regions of highly sooting (kerosene) and less sooting (methanol, acetone and ferrocene-doped kerosene) pool fires are examined and compared. Flow visualization is combined with mean and rms temperature and velocity measurements to chart the overall development of the fire flow fields. In addition, the nature and distribution of the soot is examined throughout the continuous and intermittent flame zones of 6.5 and 31 cm diameter pool fires.

The Experiments

The fires were established in the existing University of Waterloo pool fire test facility, which has been described in detail in reference 8. Velocities were measured (where possible) using two component LDA operated in the forward scatter configuration, while temperatures were measured with a 50 \( \mu \text{m} \) diameter bare wire Pt/Pt-10\%Rh thermocouple. Soot loading was estimated gravimetrically, and the nature of the particles examined by collecting samples onto small discs and using both optical and scanning electron microscopy with image analysis equipment.

Preliminary Results and Discussion

The overall development of the mean temperature and velocity fields in all of the fires are very similar to those that have been reported previously [8]; however, details of the distribution of temperature, and therefore the velocity field, do exhibit some variations in the fires with ferrocene added. The overall structure and visible flame height of small kerosene fires appeared unchanged with and without ferrocene added to the fuel; however, generally higher centreline temperatures were observed in small kerosene fires doped with ferrocene. In the larger kerosene fires, on the other hand, the visible flame heights appeared shortened when ferrocene was added to the fuel and the fire changed in colour and shape, indicative of the changes in soot formation and therefore radiation distribution within the flow field.

Reported supported through individual NSERC Operating Grant to E.J. Weckman
In general, the soot collected from kerosene fires with ferrocene added was much finer than that collected in the base kerosene fires. Similarly, there appeared to be higher soot particle loadings in the lower regions of the fire. Both these observations are consistent with previous work \([3, 4]\) and the theory that soot agglomeration processes are hindered by the addition of the additive.

Ferrocene added to the kerosene fire reduced total soot emission, as measured in the upper regions of the smaller fires, by about 30\%. This is of the order suggested in reference 2 for alkali earth additives in polymer diffusion flames, but less than the amount predicted in reference 6 for ferrocene additions to crude oil fires. This discrepancy is still under investigation.

**Conclusions**

Results of this study provide the first quantitative data on the effects of ferrocene addition for soot suppression in small and medium-scale open pool fires. Addition of ferrocene does not affect the overall development of the fire flow field; however, there is an effect on the details of the temperature and velocity fields.

Soot collected in the fires with ferrocene added is finer and the soot particle loading is higher near the base of the fires. In the upper regions of the fires, total soot emission has been reduced, by about 30\% for small kerosene fires.

In the smaller fires, the addition of ferrocene, and consequent reduction in soot loading, does not appear to affect either the flame height or visible structure of the fire. In the larger diameter fires, on the other hand, flame heights are shortened and the fire appears less luminous with ferrocene addition.

**References**

Modeling the Growth of Polynuclear Aromatic Hydrocarbons in Diffusion Flames

J.H. Miller
Department of Chemistry
The George Washington University
Washington, DC 20052

We have developed a model which describes the physical and chemical processes that control \( \text{PAH} \) and soot concentrations along stream lines in laminar methane/air and ethylene/air diffusion flames. The development is similar in spirit to that presented in a recent paper by Kennedy et al. [1]. In both models, soot processes are divided into three broad classifications: inception, chemical and physical growth (agglomeration), and oxidation. Both models assume that the rates of the soot processes depend on mixture fraction. In our model, we attempt to rationalize the choice of these rates from a knowledge of the fundamental chemical processes which are occurring in the flame combined with detailed measurements of species concentrations.

**Growth**

The growth chemistry in our model is based on the work of Frenklach et al. who proposed a model for ring growth based on the successive addition of acetylene to a growing aromatic radical core [2,3]. In Frenklach's simplified mechanism parent \( \text{PAH}(A_i) \) are converted into a phenyl-like radical by hydrogen abstraction. The resulting radical reacts with acetylene to form a radical addition product. A subsequent acetylene addition irreversibly forms the next largest parent \( \text{PAH} \). In this reaction scheme, using steady-state estimates for the concentrations of phenyl and phenylacetylene radicals, the rate of formation of the \( A_{i+1} \) \( \text{PAH} \) can be written as

\[
\frac{d[A_{i+1}]}{dt} = k_{\text{eff}} [A_i],
\]

where \( k_{\text{eff}} \) is dependent upon the concentrations of \( \text{C}_2\text{H}_2 \), \( \text{H}_2 \), and \( \text{H} \). In addition to this chemical growth process, agglomeration of \( \text{PAH} \) is also considered [4].

**Inception**

As seen above, aromatic growth is treated as an irreversible sequence of acetylenic addition steps to a growing aromatic core whose initial source is the one ring species \( A \) (i.e., benzene). Although, there have been a number of models proposed for the chemistry of the formation process of benzene (invoking either reactions of \( \text{C}_2 \) with \( \text{C}_4 \) species [2] or the reaction of two \( \text{C}_3 \) species [5]), no clear consensus exists for the dominant channel in hydrocarbon flames. Because of this uncertainty, we will adopt a different approach for deriving an inception rate. Over the past few years an extensive data base for species concentrations, temperature, and velocity in a laminar methane/air diffusion flame has been collected. These data have included measurements of the concentration of stable species such as acetylene, benzene, and molecular hydrogen [6] and has been recently extended to include profile measurements of radical species [7,8]. With this data base, we have demonstrated how the overall rates for a species' chemistry may be calculated by solving its conservation equation directly. For the work presented here, we will use the correlation of the benzene formation rate with mixture fraction as calculated using this technique [9]. For mixture fractions where this net rate is negative (such as at the high temperature reaction zone), the inception rate will be set to zero.

**Oxidation**
In combustion systems, Soot and PAH may be oxidized by a number of species with 0-atoms, molecular oxygen, and hydroxyl radical believed to be dominant. In the model we include both hydroxyl radical reactions with parent PAH and the reaction of molecular oxygen with radical PAH.

**Mixture Fraction Libraries**

Concentration profiles for $\text{C}_2\text{H}_2$, $\text{H}_2$, $\text{H}_\text{i}$, OH, O, temperature, and mixture fraction (derived from these and other species profiles) collected in a laminar methane/air diffusion flame supported on a Wolfhard-Parker burner were used to build a library of growth and oxidation species. With the exception of $\text{C}_2\text{H}_2$, the concentrations of these species showed little variation with flame height. Acetylene concentrations grow with height for the first 9 mm above the burner surface and then appear to level off. All profiles used in developing the library were collected at this 9 mm height. Much less chemical data is available for ethylene/air diffusion flames. Therefore, estimates based on equilibrium (full, super, and partial) must be made for the radical species concentrations.

**Flame Field Calculations**

We have modeled the structures of both methane/air and ethylene/air diffusion flames which have been studied by Santoro et al. [10]. Velocity, density, and mixture fraction fields throughout the flame were obtained using a computer code which has been described previously [11-13].

**Results and Discussion**

Results for both methane/air and ethylene/air flames will be presented. These results will be compared with experimental measurements of soot volume fractions, carbon monoxide concentrations, and flame temperature [14].

**References**

AEROSOL DYNAMIC PROCESSES OF SOOT AGGREGATES IN A LAMINAR ETHENE DIFFUSION FLAME

R. Puri, T.F. Richardson, and R.J. Santoro
Department of Mechanical Engineering
Pennsylvania State University
University Park, PA 16802
and
R.A. Dobbins
Division of Engineering
Brown University
Providence, RI 02912

Introduction

Soot particles consist of aggregates of a variable number of smaller primary particles which are uniform in size and more or less spherical in shape. The optical cross sections of these polydisperse aggregates have been formulated [1] and were used by Dobbins et al. [2] to reduce laser light scattering/extinction measurements in an ethene/air diffusion flame. In the present work, the intensity of the scattered light was measured simultaneously at three angles - 45°, 90° and 135° rather than sequentially [2]. This improved data allows the determination of the fractal dimension D, by an iterative procedure. The new data (in conjunction with existing information on the local extinction, the primary particle size and velocity field information) allow a more detailed analysis of the observations. The current analysis provides a global description of the physical processes associated with formation, growth and oxidation of soot aggregates.

Experimental

A coaxial burner [3], having a 1.1 cm diameter fuel tube and a concentric 10.2 cm diameter air annulus, was supplied with ethene and air at flow rates of 3.85 and 713 cm³/s respectively. The vertically polarized laser beam from an Argon ion laser having a wavelength of 514.5 nm and incident power of 0.5 W was modulated at 1015 Hz before being focussed onto the flame. Scattered light receivers (comprising a polarizer, a 12.7 mm circular aperture, collection lens, a 1 mm pinhole aperture, laser line filter and a photomultiplier tube) were located at 45°, 90° and 135°. The detectors were connected to lockin amplifiers and to a computer employing a GPIB interface; thus providing near simultaneous data acquisition from each detector. The detectors were calibrated using a Rayleigh scattering technique [3].

Results and Discussion

Analysis of the light scattering data, when combined with Transmission Electron Microscopy (TEM), employing an approach previously reported by Dobbins et al. [2] yields information on the aggregate size, primary particle size, aggregate number concentration, aggregate surface area, and average number of primary particles per aggregate. The light scattering results indicate that the aggregates can be described as fractal-like [4] with a fractal dimension of D, = 1.74. Figure 1 shows the measured results obtained for the soot volume fraction, f, the primary particle diameter, d_p (as determined from TEM images) and volume
mean diameter, $D_{\text{mean}}$ of the aggregates. These three curves show maximum values at about 56 ms which corresponds to a height in the flame of 50 mm from the burner exit. This point marks the transition from the growth region, where surface growth and cluster-cluster aggregation are contributing to particle size increases, to the oxidation region where oxidation reactions oppose aggregate growth. For this same region of the flame, the number concentration of the primary particles remains constant while the number concentration of aggregates monotonically decreases with time (or height in the flame). This aggregation process occurs at a rate similar in magnitude to theoretical calculations for the coagulation of spherical particles in the free molecular limit. The data reduction of the soot particle field that is based on the presence of aggregate morphology presents an internally consistent picture of the development of the soot particle field in laminar diffusion flames.

Acknowledgement

This research was performed under the sponsorship of the U.S. Department of Commerce, National Institute of Standards and Technology, Center for Fire Research under grant number 60NANB0D1035 to the Pennsylvania State University and grant numbers 60NANB9D0975 and 60NANB1D1110 to Brown University.

References

Carbon Monoxide and Soot Particle Formation Studies in Underventilated Laminar Diffusion Flames.

S.G. Léonard, R. Puri and R.J. Santoro
Department of Mechanical Engineering
Pennsylvania State University
University Park, PA 16802
and
G.W. Mulholland
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

Introduction Carbon monoxide (CO) has been identified as one of the most serious products of incomplete combustion resulting from fires. Since the severity and the rapidity of its effect depend on concentration\[1\], a better understanding of its formation with respect to other species is of great interest. Studies of the fundamental process of CO formation and its relationship to soot have recently been reported \[2\]. However, the use of overventilated diffusion flames for these studies limited the range of fuel equivalence ratio which could be studied. Underventilated flames offer the opportunity to study much richer fuel conditions, but previous studies of this flame configuration are lacking. In the present study a series of underventilated laminar diffusion flames have been investigated with respect to the CO and soot yields as a function of fuel and oxidizer ratio and flow rate. Based on the analysis of soot samples collected on filters, information on the organic carbon content of the soot has also been obtained. Preliminary results for these studies are presented here to provide an overview of CO and soot formation under underventilated flame conditions.

Experimental Approach Underventilated diffusion flames refer to conditions where the amount of oxidizer flow is insufficient to result in the oxidation of all of the fuel to carbon dioxide and water. In the following study, the fuel to oxidizer ratio is characterized by the overall equivalence ratio, \( \Phi \), based on the ratio of the inlet fuel and air flow rates, where \( \Phi \) greater than one correspond to fuel rich conditions. The burner consists of two concentric tubes which could be varied in size to investigate the sensitivity of the results to burner geometry effects. In the present work, results for a single burner will be presented which are representative of the general behavior observed for the CO and soot yields. This burner had a fuel tube of 0.73 cm i.d. and outer air annulus with a 2.88 cm i.d. Both methane \(( \text{CH}_4)\) and ethene \(( \text{C}_2\text{H}_4)\) fuels were studied. For the methane flames, fuel flow rates of 10 cm\(^3\)/s and 20 cm\(^3\)/s were examined, while for ethene, flow rates of 3.2 and 6.4 cm\(^3\)/s were considered. The air flow rates were selected in each case to allow the overall equivalence ratio, \( \Phi \), to be varied between 0.5 and 4.0. Measurements of CO and CO\(_2\) concentrations were obtained using individual NDIR instruments while soot particles were collected separately using filters and analyzed for mass deposited by a weighing procedure. Yields for CO and soot on a mass basis were determined from the measured concentrations and known flow rates of the gases supplied to the burner. Samples were withdrawn near the top of a glass chimney enclosure at a location approximately 50 cm above the burner exit. A large nitrogen dilution flow of 472 cm\(^3\)/s was introduced in a mixing section of the enclosure to cool the product gases and aid in achieving uniform concentration profiles across the exit where sampling occurred. A tripper plate was placed at the connection between the main burner and the mixing section to assist in this process.

Results and Discussion Figure 1 shows the CO and soot yields as a function of equivalence ratio for the ethene flame with a fuel flow rate of 6.4 cm\(^3\)/s. It should be noted that this fuel flow rate corresponds to conditions well above the smoke point for ethene \[3\]. For these conditions, the soot yield is observed to initially increase slightly as \( \Phi \) is increased from one-half. However this quantity then is observed to decrease for \( \Phi \) greater than one with the soot yield varying by about one order of magnitude as \( \Phi \) is varied from lean to rich conditions. For \( \Phi \) greater than one, CO is qualitatively observed to follow a similar although somewhat smaller variation. For lean overall equivalence ratios \((0.5 \leq \Phi < 1)\) the yields of CO are observed to be quite low. These trends are consistent with previous observations regarding the behavior of soot and CO in overventilated laminar diffusion flames. Analysis of the elemental and organic carbon content of soot formed in these underventilated flames reveals for \( \Phi \geq 2 \) that the organic to elemental carbon ratio exceeded one for both the methane and ethene flames. Such
large amounts of organic carbon in the soot particles is viewed to be quite interesting and differs from observations for soot formed in overventilated flames where the organic carbon content is usually much lower.

Future work will emphasize more detailed comparisons with overventilated flame results as well as further analysis of the fuel structure and burner geometry effects.

![Graph](https://via.placeholder.com/150)

Figure 1. Yield of CO (+) and soot (+) for the ethene flame at a fuel flow rate of $6.4 \text{ cm}^3/\text{s}$. Yield of CO is in grams of CO per gram of fuel ($\text{C}_2\text{H}_4$) and yield of soot is in grams of C per gram of fuel. Measurements of CO are based on a dry analysis. The lines of the graph are only to indicate trends and are not curve fits.

Acknowledgement The work done at the Pennsylvania State University was supported under grant 60NANB0D1035 from the National Institute of Standards and Technology.

References
A MODEL OF SOOT WITH DETAILED CHEMICAL KINETICS
Clement Yam and Ian Kennedy
Department of Mechanical, Aeronautical and Materials Engineering,
University of California
Davis CA 95616

Introduction. Thermal radiation from fires is an important aspect of fire spread and ignition. Fires are usually highly luminous i.e., much of the radiation is due to the presence of particles of carbonaceous soot in the flames. The prediction of luminous radiation continues to be a difficult problem in combustion research because of the inherent difficulty in predicting the amount of soot that is present in a flame. Several approaches to the prediction of soot and its radiation have been explored over the last decade. Some methods rely on scaling approaches or on empirical correlations of radiation while others attempt to solve conservation equations for the amount of soot in a flame. Among the latter approaches there are two methods. One method attempts to predict the production of soot with detailed chemical kinetics from acetylene to the first aromatic ring and upwards in molecular weight. This approach is strictly a research tool; it will not be possible to implement the complex kinetics in an engineering prediction in the near future. The other possible method is a semi-empirical approach in which knowledge of the soot process that is derived from experiments is used to develop a correlation of soot production and oxidation rates. The rates are then used in conservation equations that are solved numerically. This semi-empirical approach to soot formation and oxidation has been explored in this project.

Model for Soot Formation and Oxidation The basis for the model of soot that we have developed is a correlation between soot surface growth rates and the local equivalence ratio in a flame i.e., the mixture fraction. This correlation was obtained from experiments in a counterflow diffusion flame. An Arrhenius dependence on temperature is also assumed for the surface growth rate. The rate of growth is assumed to depend on the total surface area of the soot aerosol. It was found that the details of the nucleation step were not necessary in order to achieve satisfactory predictions of soot volume fractions. Rapid coagulation of freshly nucleated particles collapsed the number of particles to low values and there was a resulting insensitivity to the nucleation step when soot volume fractions were high. Therefore, calculations were performed with only the soot volume fraction equation as part of an integrated flow field prediction. The other equations that were solved included the continuity equation, the momentum equation and an energy equation. The latter equation included a radiative heat loss term to account for the energy lost by luminous radiation from soot.

The equation for the soot volume fraction included two source terms and two sink terms. The source terms were the contribution to the soot volume fraction due to the nucleation of new material and the formation of soot due to surface growth. The sink terms included oxidation due to attack by OH and by O_2. Equilibrium values of OH and O_2 at the local mixture fraction and enthalpy were used in the original predictions that are reported in detail in the 24th Symposium on Combustion.

Recent measurements by Pun et al. of OH concentrations in a sooty ethylene flame have raised doubts about the validity and accuracy of the modeling that has been used for the oxidation step in the soot process. The LIF measurements indicate that OH concentrations are up to an order of magnitude higher than the equilibrium values that have been used in
the soot modeling. Furthermore, the model that we have developed has shown that the loss rate of OH on soot is comparable to the estimated net production rate of OH in the flame. Therefore, a goal of the project has been the relaxation of the equilibrium assumption regarding OH (and other radicals such as O). The impact of soot burnout on the flame chemistry and the inverse problem of the impact of finite rate OH kinetics on soot oxidation have been investigated.

Calculations with Detailed Chemistry. A model has been developed for the co-flow axisymmetric diffusion flame. The governing equations were solved in boundary layer form. Hence, the parabolic system of pde's could be solved with considerably greater speed than can be achieved if the full elliptic problem is solved. For example, the calculations that are presented here were obtained on a UNIX workstation in about 7 hours; the elliptic problem requires many hours on a Cray. Neglecting phenomena such as diffusion in the axial direction is unlikely to be important in the present situation in view of the inaccuracies surrounding the soot model. A detailed kinetic scheme for methane and air has been used. The "short" mechanism from Sandia with forty six reactions was adopted. Seventeen species equations were solved along with the momentum equation, the energy equation and the soot volume fraction equation with the same source terms as before. However, in the present treatment of oxidation, the actual predicted OH and O₂ mole fractions are used. In addition, a sink term in the equation for OH has been included in order to model the loss of OH on soot particles. In a sense, the model as it stands at present is a hybrid model with detailed chemistry for methane but surface growth rates that apply to an ethylene flame. In the near future, C₂ chemistry will be incorporated into the model so that true ethylene-air chemistry can be treated.

Predictions of Soot, OH and O with Finite Rate Kinetics. The axisymmetric ethylene diffusion flames of Santoro et al have been calculated. The amount of soot increases along each of the flames until oxidation becomes important towards the end and the soot volume fractions begin to decrease. The soot layer lies within the flame envelope. The peak soot distribution is in reasonable agreement with the levels that were reported by Santoro et al from laser extinction measurements.

Equilibrium values of OH were obtained by running STANJAN with the calculated temperature at each radial point and the calculated N, O, C, and H atom ratios. Large superequilibrium values of OH are predicted. At the peak OH location the calculated OH mole fraction can exceed the equilibrium value by as much as a factor of 10. This value is in accord with the measurements of Puri, Moser, Santoro and Smyth in an ethylene flame. Further work needs to be done with ethylene kinetics to be certain of this result but it is not likely to be greatly different. The peak OH mole fraction decreases along the flame, primarily as a result of the drop in temperature due to radiation.

Another potentially important oxidant of soot is the O atom. Large superequilibrium concentrations are obtained, in this case up to a factor of 60 larger than equilibrium. The soot model does not include O as an oxidant at present. These results suggest that it may be potentially important. The assumption of equilibrium concentrations for OH are also inaccurate. It is not clear at present how these non-equilibrium effects can be incorporated into the basic soot model whilst retaining its potential for use in turbulent flame calculations.

Acknowledgement. This work was performed with the support of NIST through Grant 60NANBOD1055.
THE OXIDATION OF CARBON MONOXIDE AND SOOT IN HYDROCARBON DIFFUSION FLAMES

Rahul Puri, Marlow Moser and Robert J. Santoro
Department of Mechanical Engineering
Pennsylvania State University
University Park, PA 16802
and
Kermit C. Smyth
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

Introduction Since CO inhalation is one of the major causes of fire fatalities, significant effort has been directed towards obtaining an understanding of the increased CO production in fires. Numerous workers have observed a correlation between the amounts of CO and soot produced in diffusion flames [1] as well as compartment fires [2].

Our earlier work [3] sought to obtain a fundamental understanding of the relationship between the production rates of CO and soot in diffusion flames. The effects on the CO depletion rates of lower temperatures (due to radiation heat loss from the soot particles) and competition between CO and soot for the oxidizer species OH· were examined using equilibrium estimates of the OH· concentration. Based on the rates of CO and soot oxidation due to OH·, competition between these species for OH· was found to be plausible in both fuel lean and fuel rich regions of the flames studied. While radiation quenching (through reduction of OH· radicals) seemed important in the fuel lean regions, the equilibrium concentrations of OH· did not correlate with the observed CO destruction rates in the fuel rich regions.

Our recent work [4] has shown that equilibrium estimates of OH· concentrations are not reliable, with the departure from equilibrium depending upon the soot loading. Therefore, in order to carefully evaluate the interaction between CO and soot, OH· concentration measurements were extended to the flames of our earlier study [3].

Experimental Approach Laminar diffusion flames burning fuel mixtures were used to vary the soot concentration while keeping the total carbon flow rate constant. Flames burning methane (9.8 cm³/s) and methane (5.6 cm³/s) doped with either butane (1.05 cm³/s) or 1-butene (1.05 cm³/s) were studied. The OH· concentrations were measured in the soot oxidation region of these flames using laser-induced fluorescence. The experimental arrangement and the measurement procedures are described in detail elsewhere [4].

Results And Analysis Figure 1 presents the OH· concentration as a function of the axial centerline position in the three flames, with the error bars showing the estimated uncertainty. The pure methane flame and the methane/butane flames have maximum centerline soot volume fractions of 0.76 and 2.55 ppm respectively and completely oxidize all the soot produced within the flame. Consequently these flames have a closed flame tip at axial locations of 10.7 and 11.7 cm respectively. The methane/1-butene flame has a maximum centerline soot volume fraction of 5.46 ppm and does not have a closed flame tip since it emits smoke. This figure clearly shows that as the soot concentration increases, the centerline OH· concentration decreases. The most noteworthy feature is that the pure methane and the methane/butane flames have approximately the same OH· concentration along the flame centerline once the soot particles are completely oxidized. Thus, the decrease in the OH· concentration in the presence of soot particles can be attributed to chemical consumption by soot particles.
In addition to the soot concentration, the OH concentration can be expected to depend on the local stoichiometry (\(\phi\)), temperature (T) and the CO concentration (which may not be uniquely related to \(\phi\) [3]). Equilibrium calculations reveal that the OH concentration is not very sensitive to \(\phi\) for \(\phi \leq 0.95\). The lean flame region, with the elimination of the \(\phi\) dependency, is amenable to a simple analysis for the evaluation of the relative contributions of thermal and chemical effects to the observed OH concentration.

Along the centerline the OH concentration depends upon the temperature and the concentrations of CO and soot, i.e. \([\text{OH}^\cdot] = f(T, [\text{Soot}], [\text{CO}])\). However, on the lean side of the flame front, where there is no soot and little CO, the OH concentration depends only on the temperature, i.e. \([\text{OH}^\cdot] = f(T)\). A comparison for flames with different soot concentrations allows a simple estimate to be made of the relative contributions of thermal and chemical effects on the CO depletion rates. If

\[
F_1 = \frac{f(T, \text{Butane})}{f(T, \text{1-Butene})}; \quad F_2 = \frac{f(T, [\text{Soot}], [\text{CO}]) \text{Butane}}{f(T, [\text{Soot}], [\text{CO}]) \text{1-Butene}}
\]

then the relative contribution of radiative quenching to the observed decrease of the OH concentration in the methane/1-butene flame is the ratio \(F_1/F_2\). This analysis indicates that radiative quenching makes only a 6% ± 2% contribution to the observed lower OH concentration along the centerline in the methane/1-butene flame. Thus, the OH concentration in the presence of soot particles is suppressed primarily due to oxidation reactions as opposed to thermal quenching effects.

**Acknowledgement.** The work done at the Pennsylvania State University was supported under grant 60NANB0D1035 from the National Institute of Standards and Technology.

**References**

Measurements of Carbon Monoxide Concentration and Temperature in Axially-Symmetric Methane and Ethylene Diffusion Flames

J.H. Miller, Reed Skaggs, and Bijan Ahvazi
Department of Chemistry
The George Washington University
Washington, DC 20052

Carbon monoxide is an important flame species. A substantial fraction of the heat release in a flame occurs from the CO oxidation reaction:

\[ \text{CO} + \text{OH} \rightarrow \text{CO}_2 + \text{H}_2 \]

In many combustion systems this reaction is essentially irreversible, which serves as the basis for the development of global oxidation kinetics relationships of the form [1]:

\[ \frac{d[\text{CO}]}{dt} = -k_{ov}[\text{CO}][\text{O}_2]^{0.25}[\text{H}_2\text{O}]^{0.5} \]

where \( k_{ov} \) represents the effective, overall rate constant for the process. In other combustion systems, this reaction is close to equilibration, which may explain the correlation of carbon monoxide concentration with local equivalence ratio (or, alternatively, mixture fraction) [2].

Because of the effectiveness of its binding with hemoglobin, carbon monoxide is largely responsible for the loss of life in fire situations [3]. In efforts aimed at the development of a predicative capability for carbon monoxide emissions from fires, various workers have found that CO concentrations can be correlated against overall stoichiometry in the ceiling layers of room fires. Of course, although similar in overall appearance, these correlations are quantitatively different from those observed within the flame itself [2]. Because the carbon monoxide oxidation reaction may effectively be frozen below a certain temperature (due to a decrease in hydroxyl radical concentrations), the conceptual link between in-flame and global correlations will be established via a coupling of a carbon monoxide oxidation model with a turbulence code capable of predicting the occurrence of this thermal quenching. Although this task may be far into the future, the data required to validate such a model must come from measurements of CO concentrations in both laminar and unsteady diffusion flame conditions. This will eventually require not only quantitative accuracy, but also fast temporal response, as can be provided by tunable diode laser diagnostics.

The emission of soot particles from flames is likely to also affect the concentration of emitted carbon monoxide. There are two reasons for this assertion. First, large soot particles radiate considerable energy, thus cooling the local flame environment. If the cooling is substantive enough, the rate of carbon monoxide oxidation will decrease. Secondly, copious amounts of carbon monoxide accompany the oxidative pyrolysis of aromatic compounds [4]. Since soot has essentially an aromatic structure, carbon monoxide might be an expected product of the soot oxidation process.

Measurements of carbon monoxide concentrations in Wolfhard/Parker diffusion flame systems have been accomplished in three ways: a laser-induced fluorescence technique [5], which gives only a relative determination of concentration, a quartz microprobe/mass spectrometric technique [6], and tunable diode laser absorption spectroscopy (TDLAS) [8]. For the mass spectrometric technique, profiles of signal at molecular weight 28 were collected at three different ionization energies. In this flame, there are three species which occur at this mass to charge ratio: CO, N2, and ethylene. Because carbon monoxide has a slightly higher ionization potential than ethylene and a slightly lower one than nitrogen, CO concentrations can be calculated by subtraction of profiles combined with the application of molecule and ionization-energy specific calibration factors. This is a somewhat
cumbersome process and is appropriate only for laminar flames. It should also be noted that this technique is only possible when soot concentrations are quite low because of clogging of the microprobe orifice by particles [7,9]. Because light scattering intensity is proportional to \( \lambda^4 \), absorption spectroscopy in the mid-infrared spectral region is free from interference by particulates. Thus, the quantitative determination of carbon monoxide concentration in the presence of soot is most easily accomplished with TDLAS.

Much of the available data on soot concentrations in diffusion flames has been collected in axially symmetric jet burners [10]. In the work described here, we apply TDLAS to measurements of both carbon monoxide concentration and local flame temperature in axial ethylene/air and methane/air diffusion flames. Because TDLAS is a line-of-sight technique, tomographic reconstruction of the flame structure is required to obtain radial profiles [1-14]. The results of these measurements are compared with computed flame structures and soot concentration maps for these flame systems [15].

References

Acknowledgment

This work was supported through grants from the Building and Fire Research Laboratory of the National Institute of Standards and Technology and The National Science Foundation.
Introduction

Recent studies attribute a large percentage of fire injuries and deaths to the products of incomplete combustion such as smoke and carbon monoxide (CO). Carbon monoxide has been implicated as the major toxicant by investigations [1-3] which indicate that more than half of all fire victims have fatal levels of carboxyhemoglobin in their blood. In an effort to understand and predict how CO is produced in enclosure fires, a long-term project [4] has been implemented within the Building and Fire Research Laboratory. The findings of the study will be used to create more realistic fire models and in the development of strategies for reducing the number of deaths attributed to CO. This paper describes the construction and operation of a reduced-scale enclosure (RSE) fire facility. This enclosure was designed to model important physical phenomena of a fire located in a compartment. Natural gas was the fuel source for over 140 fires with heat release rates (HRR) ranging from 7 to 650 kW. Fires of greater than 150 kW HRR created post-flashover conditions within the RSE. Temperatures inside the RSE and CO, CO₂, and O, concentrations inside and outside the RSE were measured at different elevations and positions for each fire.

Experimental

The RSE is a 40% model of the standard room proposed in ISO/DIS 9705 [5]. The RSE internal dimensions are 0.98 m wide by 0.98 m tall by 1.46 m deep. A 0.48 m wide by 0.81 m tall door is centered at the bottom of the front wall of the enclosure. The enclosure consists of a steel frame with sheet metal sides and is lined with two layers of 1.27 cm thick calcium-silicate board. The RSE is instrumented with thermocouple trees located in the front and rear. Cooled and uncooled probes were positioned at different locations to sample the upper combustion layer. CO and CO₂ concentrations (NDIR analyzers) and O, concentrations (paramagnetic analyzers) were measured. Attempts were made to measure air influx through the door with a pressure probe and aspirated thermocouples located in the doorway.

The air flow in conjunction with fuel flow data was used to estimate a global equivalence ratio, $\phi_g$, for the enclosure. $\phi_g$ is defined as the ratio of the mass of fuel-derived products to the mass of air-derived products in the upper layer normalized by the stoichiometric fuel-to-air ratio. Other researchers [6-9] have proposed using $\phi_g$ to correlate major species concentrations, including CO, in the upper layer of a fire.

Typically, the burner was centered in the enclosure and each fire ranged from 15 to 20 minutes in duration. Different burner positions and a narrow door configuration were used in a few fires to study the effects on flow patterns and transient behavior, respectively. Several horizontal and vertical mappings, where a sample probe was moved during a fire, were also performed to characterize the species concentration profiles within the enclosure. Since water interferes with NDIR measurements, it was trapped out of the sample gases and estimates of water concentrations were calculated. This allowed the data to be reported on a wet volume basis.

Results and Discussion

Figure 1 shows a plot of average steady-state CO concentrations for each fire versus the heat release rate of the fires. Figure 2 shows the corresponding O, concentrations. Due to difficulties in measuring $\phi_g$, the results are presented versus HRR. For reference purposes, a 200 kW fire had a corresponding $\phi_g$ of approximately 1. At 100 kW HRR, the CO concentration begins to rise and levels off to 3 % in the front and 2 % in the rear for fires with greater than 400 kW HRR. Hood experiments by previous investigators [6-8] yielded 1-2 % CO concentrations for similar $\phi_g$. The O, concentration drops to near zero at 200 kW and remains near zero for all higher HRR. CO concentrations are higher in the front than in the rear of the enclosure. Conversely, CO₂ concentrations are higher in the rear and lower in the front. Temperatures in the front of the enclosure during the high HRR fires are consistently 200 - 300 °C higher than in the rear. Cooled and uncooled probe concentration comparisons at the same locations were nearly identical suggesting no reactions occurred in the probes.

A companion field modeling of RSE fires [10] suggests that flow patterns within the enclosure introduce oxygen directly into the upper layer in regions between the doorway and the fire. For the high
APPLICABILITY OF THE GLOBAL EQUIVALENCE RATIO FOR PREDICTING CARBON MONOXIDE IN ENCLOSURE FIRES

William M. Pitts

Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

Roughly two thirds of all deaths resulting from enclosure fires can be attributed to the presence of carbon monoxide (CO) [1,2] which is known to be the dominant toxicant in fire deaths [3]. The mechanisms responsible for the generation of high concentrations of CO are poorly understood. A long-term program (Carbon Monoxide Production and Prediction Project) at the Building and Fire Research Laboratory is seeking to develop an understanding of and predictive capability for the generation of CO in fires. [4] The vast majority of this research effort during the past four years has focused on the following question:

Can the generation behavior of CO observed in hood experiments designed to model two-layer burning be extended to predict CO generation in actual enclosure fires?

It has been found that the major products of combustion measured in hoods located above open fires in the laboratory are strongly correlated with the global equivalence ratio (defined as the ratio of masses derived from fuel and air normalized by the ratio for stoichiometric burning) for the gases trapped in the hood. [5,6] These correlations are quite robust, holding even when air is injected directly into the hood instead of entering only through the fire plume. [6] The existence of these correlations has been termed the Global Equivalence Ratio (GER) concept. A dependence of the combustion products in the hoods on the upper-layer temperature has been noted.

Recently, investigations have been completed which provide significant insights into the mechanism of CO formation in enclosure fires. The findings allow an assessment of the conditions for which the application of the GER concept to enclosure fires is appropriate. The results of the these studies are briefly summarized and are used to answer the above question.

The reaction behavior of gases observed in the low-temperature hood experiments when the gases are heated to higher temperatures has been examined. [7] It was found that the gas mixtures, which were observed to contain both fuel and oxygen, became reactive as the temperature was raised. For lean conditions, reaction results in the formation of CO, and H₂O. For rich conditions the oxidation generates primarily CO in preference to CO₂. Due to shifts in the ratios of H₂O and H₂ produced, the calculated increases in CO are larger for temperatures in excess of 1000 K.

Roby and coworkers [8] have monitored CO formation inside a reduced-scale enclosure designed to provide separate ventilation and exhaust pathways. The design ensures the development of two well-defined layers and that entrainment into the upper layer occurs only through the fire plume. Fuels used in the study include hexane, wood, PMMA, and polyurethane. This study has confirmed a temperature effect on the combustion products observed in the upper layer, but indicates that the correlations derived from the hood experiments provide good estimates for the concentrations of upper-layer combustion gases when the temperature effect is accounted for properly.

In a separate investigation of fires within a reduced-scale enclosure (RSE), Bryner, Johnsson, and Pitts have investigated combustion gas production in a 2/5-scale model of a full-sized room. [9] Natural gas was used as the fuel with the burner centered in the room. Unlike the hood experiments [5,6] and the enclosure fire investigation of Roby et al. [8], significant variations in CO concentration were found with position in the
upper layer of the fire. In particular, for a room having a doorway scaled for fire size and ventilation relative to the full-scale enclosure, CO concentrations observed near the doorway for fires having rich upper layers were much higher than observed in the hood experiments of Toner [6] with the same fuel. Interestingly, concentrations in the rear of the enclosure were very similar to Toner's observations. Oxygen concentrations were very low (< 0.5%) at both locations in the RSE.

Limited experiments were done with a narrow doorway for the RSE in which upper-layer temperatures were very similar to those observed in the hood experiments of Morehart [6]. Upper-layer concentrations within the enclosure were more uniform and were consistent with those measured in the hood experiments.

Preliminary experiments were done where the upper walls and ceiling of the RSE were lined with 6.4 mm thick plywood. A high heat release rate natural gas fire (600 kW) was burned. Previous experiments [10] suggested that primarily carbon monoxide would be generated by wood pyrolysis for the high temperature, low oxygen conditions expected in the upper layer. Concentrations of CO as high as 14% were observed during the fire. This experiment demonstrates that pyrolysis of oxygenated polymers in the upper layer of an enclosure fire can be an important source of CO. Apparently, this mechanism of CO generation has not been recognized previously.

A field modeling investigation characterized the flow patterns within the RSE. [11] These calculations showed that air from the lower layer was entrained directly into the upper layer, i.e., the air did not pass through the fire plume. Based on the kinetic calculations [7], this air would be expected to react with rich upper-layer gases and generate CO instead of CO2. This hypothesis provides an explanation for the higher CO concentrations observed in the front of the RSE for rich, high-temperature upper layers.

The findings of the above studies allow the conditions for which the GER concept can be used to predict combustion gas generation in enclosure fires to be assessed. It is expected to be applicable in fires: 1) which are fully ventilated, 2) which have low upper-layer temperatures, and 3) which have high-temperature upper layers for which the only pathway for air to enter the upper layer is through the fire plume. It is expected to be inappropriate in fires: 1) which have rich, high-temperature upper layers and in which air can enter the upper layer directly, and 2) where solids capable of pyrolysis are located in the oxygen-depleted, high-temperature upper layer.

The enclosure fires which are responsible for most CO-induced fire deaths are flashed over and underventilated. [4] Such fires are expected to have intense mixing in which air may be entrained directly into the upper layer. Additionally, these fires often have fuels which can be pyrolyzed located in the upper layer. On this basis, it is concluded that great care should be exercised when applying the GER concept to predict CO production in actual enclosure fires.

References
[7] W. M. Pitts, Twenty-Fourth Symposium (International) on Combustion, accepted for publication.
HALON REPLACEMENT EXTINGUISHANT CONCENTRATION REQUIREMENTS: 
EXPERIMENT AND MODEL

Ronald S. Sheinson\textsuperscript{a} and Samuel P. Baldwin\textsuperscript{b}

NAVAL RESEARCH LABORATORY
Navy Technology Center for Safety & Survivability, Code 61 80
Washington, DC 20375-5320, USA, (202) 404-8101, FAX (202) 767-1716

The brominated halocarbons halon 1301 (CF\textsubscript{3}Br) and halon 1211 (CF\textsubscript{3}BrCI) are widely used as efficient, clean, non-toxic, non-conducting fire suppression agents. However, their production will cease in the near future due to stratospheric ozone layer depletion concerns. Alternatives are needed to insure safety, system performance, and mission capability.

The search for halon replacements is not new to NRL. Approximate kinetic calculations performed in 1976\textsuperscript{1} showed that brominated halons were at least as potent potential ozone layer depleters as were chlorinated CFCs, and very likely to be more so. Fire suppression research showed a physically acting agent requirement predictive model could be constructed\textsuperscript{2}. Over half the effectiveness of halon 1301 was due to bromine radical chemical reactions; iodine was equivalent to bromine in suppression efficacy; and the CF\textsubscript{3} radical itself was a chemical suppressant\textsuperscript{3}. Halogen acid generation from halon extinguished fires could range from hundreds to thousands of parts per million in concentration\textsuperscript{4,5}.

No available candidate fulfilled the requirements of being clean, non-toxic, non-conducting, and highly efficient, while also having greatly reduced ozone depleting behavior. The tremendous growth in CFC and halon usage and the now known extent of stratospheric ozone layer depletion dictate that performance compromises must be accepted.

NRL reinitiated halon replacement studies several years ago including flame suppression mechanisms and suppressant concentration requirement modeling\textsuperscript{6,7}, intermediate scale total flooding candidate evaluation\textsuperscript{8}, and atmospheric lifetime studies. We also interact directly with manufacturers through secrecy agreements and as a United States Representative member (RSS) on the United Nations Environment Program, Halons Technical Options Committee\textsuperscript{9}.

This presentation will be on laboratory burner extinction studies and resulting mechanism and modeling efforts. Other presentations from NRL at this meeting will describe our larger scale total flooding fire extinguishment studies carried out on eight candidate and model suppressants in a 56 m\textsuperscript{3} compartment at our Chesapeake Bay Detachment, Maryland, location.

Minimum extinguishment concentrations for the currently available candidate replacements using n-heptane fuel in the NRL cup burner are listed in Table I. The gas volume concentrations allow calculation of weight and liquid volume requirements relative to halon 1301. The non-ozone depleting compounds are decidedly less efficient. We have extended cup burner studies to include suppressant mixtures and varying oxygen concentrations. Halon 1301 efficiency is a non-linear function of concentration\textsuperscript{10}. A "free oxygen" model based on Ref. 11 has been developed that allows the prediction of suppressant concentration requirements in agent

\begin{itemize}
  \item Supported in part by the Office of Naval Technology and the Naval Sea System Command.
  \item \textsuperscript{a} Author to whom correspondence should be addressed.
  \item \textsuperscript{b} Current address: Johns Hopkins University, Baltimore, MD.
\end{itemize}
mixtures and blends including as a function of oxygen concentration. The additional suppressant required to protect against reflash in a post-extinguishment, carbon dioxide rich and oxygen depleted environment can also be predicted.

Suppressant action can be characterized as physical, chemical scavenging, or chemical catalytic. The physical mechanism produces linear suppression effectiveness, while catalytic reactions involving bromine and iodine containing suppressants produce greatly increasing chemical effectiveness at lower agent concentrations. Weakly catalytic action involving chlorine, or chemical scavenging reactions involving fluorine, produce small or no deviations from linear behavior. Changes in effectiveness with bromine or iodine containing suppressant concentrations can be related to changes in hydrogen atom recombination chain length.

References:
1. Bogan, D. J., Naval Research Laboratory, unpublished results.

Table I: NRL CUP BURNER, N-HEPTANE FUEL

<table>
<thead>
<tr>
<th>AGENT</th>
<th>CONC %</th>
<th>REL WT</th>
<th>REL VOL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Halon 1301</td>
<td>3.1</td>
<td>1X</td>
<td>1X</td>
</tr>
<tr>
<td>Great Lakes FM-100</td>
<td>4.1</td>
<td>1.2</td>
<td>1.0+</td>
</tr>
<tr>
<td>Great Lakes FM-200</td>
<td>6.6</td>
<td>2.4</td>
<td>2.7</td>
</tr>
<tr>
<td>Du Pont FE-25</td>
<td>8.8</td>
<td>2.3</td>
<td>2.8</td>
</tr>
<tr>
<td>Sulfur hexafluoride</td>
<td>10.6</td>
<td>3.3</td>
<td>3.0</td>
</tr>
<tr>
<td>Du Pont FE-13</td>
<td>12.0</td>
<td>1.8</td>
<td>4.0</td>
</tr>
<tr>
<td>Carbon Dioxide CO2</td>
<td>21</td>
<td>2.0</td>
<td>4.1</td>
</tr>
</tbody>
</table>
FLAME SUPPRESSION EFFICIENCIES OF NEXT GENERATION
FIRE EXTINGUISHING AGENTS

A.P. Hamins, W.L. Grosshandler
and R. G. Rehwoldt

Building and Fire Research Laboratory
NIST

A comprehensive experimental program is described which seeks to find a suitable replacement for the ozone-depleting fire suppressant, CF₃Br, for in-flight aircraft protection. Because the effectiveness of a fire suppressant is related to the nature of the fire as well as an agent's thermodynamic properties and suppression kinetics, different experimental configurations are used to simulate the threat posed by a fire in a dry bay area of the aircraft and in an engine nacelle. The four facilities listed in the table below are used to rank the relative combustion suppression efficiency of various fluorinated compounds. Preliminary measurements in the cup burner and OFDF facilities will be presented, and the experiments which are planned for the detonation tube and the turbulent jet flame will be discussed.

The cup burner consists of a 28 mm fuel cup surrounded by a 120 mm chimney. The flame extinguishment test developed in NIST TN 1278* forms the basis of the experimental method. The apparatus has been modified to burn liquid fuels, including viscous hydraulic fluids. Flames are buoyancy dominated and mildly turbulent. The critical flow rate of gaseous suppressant needed to extinguish a diffusion flame established in the cup burner is measured.

The suppression effectiveness of the agents are also determined by measuring the critical flow rate of suppressant needed to extinguish diffusion flames stabilized between counter-flowing streams of oxidizer and a vaporizing liquid fuel. Unlike the cup burner, the effect of the flow field (strain rate) on the extinction process is an independently regulated parameter in the OFDF. This facilitates a controlled comparison of the relative suppression effectiveness of the agents. The suppressant concentration in the air stream necessary to extinguish diffusion flames is measured as a function of strain rate. Preliminary measurements in the OFDF are compared to the cup burner results.

An engine nacelle fire is typically a turbulent diffusion flame stabilized behind an obstruction in a high speed flow. Jet fuel, either as a spray or prevaporized, supplies the energy. Extinguishment occurs when a certain level of agent is supplied to the air upstream of the flame, and is likely to be affected by the turbulence intensity and velocity of the flow as well as the agent concentration. A coaxial turbulent burner has been designed to evaluate the various agents. The fuel is injected on the centerline and air is brought in parallel to the fuel in an annular region, approximately 50 mm across. The mean air velocity can be varied from 10 m/s to about 50 m/s, and the turbulence intensity can be controlled with screens placed upstream of the fuel nozzle. The flame is stabilized behind a disc located at the fuel outlet, and the temperature of the disc or the air can be increased to ensure that the flame remains stable. The agents are injected impulsively into the air stream so as to be dispersed uniformly across the tube before they reach the flame. Extinction efficiency will be determined with propane and JP8 as fuels.

Dry bay protection requires that an agent respond quickly enough to suppress the transition from a deflagration to a detonation. A worst case scenario will be investigated where the fuel has completely
vaporized and is well mixed with air. The mechanism for suppression may be distinct in detonations and diffusion flames because the chemistry may take place at different pressures and because the fuel and air are premixed rather than non-premixed. Many rate constants are pressure sensitive which could enhance or diminish the importance of particular chemical reactions. Furthermore, the structure of premixed and non-premixed flames are distinct, which may have a dramatic impact on the relative suppression effectiveness of different agents. A detonation tube has been designed to evaluate the suppression effectiveness of the agent in preventing a flame from accelerating into a premixed combustible fuel/air mixture. A long, stainless steel, 50 mm diameter closed tube is fitted with an ignition source at one end and an over-pressure blow-out port at the other end. The tube can be filled with a fuel/air mixture, initially at ambient temperature and pressure. Agent will be premixed with propane and air in the downstream half of the tube, separated from the combustible region by a thin membrane. The progress of the flame front can be monitored with fast time response piezoelectric type pressure gauges and ionization gauges which are flush mounted in the tube. Suppression of a fully established advancing flame front will be tested as a function of agent concentration.

Table. Flame Extinction Measurements

1. Cup Burner
   Fuels: JP5, JP8, hydraulic fluids 5606 and 83282
   Agents: eleven fluorocarbons, plus sodium bicarbonate

2. Opposed-flow diffusion flame
   Fuels: heptane, JP8
   Agents: eleven fluorocarbons, plus sodium bicarbonate
   Experimental variables: flame strain rate, agent concentration

3. Turbulent jet burner
   Fuels: propane, JP8
   Agents: eleven fluorocarbons
   Experimental variables: turbulence intensity, agent concentration, velocity, air temperature

4. Detonation tube
   Fuel: propane
   Agents: eleven fluorocarbons
   Experimental variables: fuel/air ratio, wave strength, agent concentration

Flame Structure of Inhibited Counterflowing Flames

I.K. Puri
Department of Mechanical Engineering, University of Illinois at Chicago
Chicago, IL 60680

A. Hamins
National Institute of Standards and Technology
Gaithersburg, MD 20899

Recent attention has been focused on determining replacements for the ozone depleting halogenated fire suppressants currently in use. Before new agents can be designed, an improved understanding of the mechanism of halogenated compounds in non-premixed flames is necessary. Although a number of experimental studies have been conducted on the suppression effectiveness of halogenated compounds in hydrocarbon diffusion flames, few studies have considered changes in the detailed flame structure due to the introduction of these compounds.

Numerical simulations of the structure of atmospheric counterflowing methane-air diffusion flames are conducted using a previously developed code [1]. The code allows calculations of stretched laminar flames using a detailed model of molecular transport of chemical species, as well as detailed mechanisms of chemical reactions. The local temperature, velocity and species concentration distributions in the flame are calculated. In this study, attention is focussed on the differences in flame structure when either an inert (N$_2$), a very simple halogenated species (CH$_3$Cl) or methane (CH$_4$) itself is added to the oxidizer stream.

The chemical kinetic mechanism employed is a combination of chemical kinetic models for methane [2] and methyl chloride [3] oxidation which contains 179 reversible chemical reactions for 38 chemical species and include C, chemistry. The chemical mechanism has been validated by comparison with the measured speeds of a number of premixed CH$_3$Cl/CH$_4$/air flames [4].

All calculations are conducted for strain rates in the non-combusting flow field equal to 50 s$^{-1}$, which represents weakly stretched flames typical of buoyancy dominated combustion situations such as fires. Computations were conducted for 5 mole % agent (N$_2$, CH$_3$Cl or CH$_4$) added to the oxidizer stream.

The effects of an inert such as nitrogen when added to the oxidizer stream are two-fold. The inert behaves as a heat sink, reducing the temperature of the products of combustion and as a diluent reducing the collision probability of chemical reactions. In contrast to the inert, the effect of CH$_3$Cl is both thermal and chemical. For additions of 5% CH$_3$Cl to the oxidizer stream, the calculated maximum temperature increases nearly 200 K. The chemical effect of CH$_3$Cl additions are evident through differences in the concentration of the radical pool. The calculated peak concentrations of O and H atoms decrease by nearly a factor of 2 for additions of 5% CH$_3$Cl to the oxidizer stream while the peak OH concentration decreases by approximately 30%. The decrease in
the radical pool is associated with the scavenging of H atom by Cl to form HCl. Chlorinated species other than HCl and Cl atom are negligible (e.g. Cl₂ < 1 ppm). The peak HCl concentration is nearly 5 mole % and is located on the oxidizer side of the temperature maximum.

The addition of 5% CH₄ to the oxidizer stream yields a lean premixed flame flowing into a diffusion flame. The maximum flame temperature increases with CH₄ addition to the oxidizer as do the peak concentrations of the radical pool.

Through a comparison of flame structures, the inhibiting effects of Cl on the combustion process will be discussed. A sensitivity analysis examining the importance of suppressant heat capacity, thermal conductivity, and the chemical reactions will also be discussed.

References


Concentration Profiles of Extinguishants in a Fire Environment;
Gas Sampling and Analysis


NAVAL RESEARCH LABORATORY
Navy Technology Center for Safety and Survivability, Code 6180
Washington, DC 20375-5320 USA, (202) 767-2766, Fax (202) 767-1716

The Naval Research Laboratory (NRL) halon 1301 total flooding extinguishant replacement program tests have been conducted in a 56 m³ (2000 cubic foot) cinderblock and concrete compartment approximately 4 x 3.4 x 4.3 m (13 x 11 x 14 feet) at our Chesapeake Bay Detachment. Special care had to be exercised to assure valid comparisons between suppressants’ as both their thermodynamic and chemical properties differ significantly. Roof vents and a door are employed to ensure proper preburn ventilation. The compartment is instrumented with thermocouple arrays, radiometers, calorimeters, and video recorders including infrared video. Pressure transducers and thermocouples are installed at several discharge system points, including at the overhead 3 m high (10 ft) nozzles.

Continuous gas samples were obtained from four compartment heights and analyzed for oxygen, carbon dioxide, carbon monoxide, and agent. Grab gas samples were taken at different locations in the compartment including near the seat of the fire at various times to obtain an understanding of agent addition hydrodynamics.

The overhead nozzles were used to dispense precalculated pressurized agent concentration onto a burning fire pan, 30.5 x 76 cm (12 x 30 in), of n-heptane positioned 30 cm (1 ft) above the floor, or into an n-heptane spray fire of the same energy output. Discharge was from a tank pressurized with nitrogen to 600 psig.


1. Author to whom correspondence should be addressed.

Supported in part by the Naval Sea System Command.
We observed significant concentration variation both as a function of location and time immediately following discharge. Therefore a concerted analysis was conducted. Agent profile concentrations were obtained by using evacuated 1.7 (100 in") stainless steel bottles. Five bottles each were connected to two cam actuated timers. One set of bottles was positioned to sample near the fire, the other, 3.7 m (12 ft) overhead. The timers were actuated at agent dump time and the bottles were filled sequentially over a 30 second period.

Bottle sample pressure was typically less than 400 torr. A subambient pressure analytical sampling system was employed to extract the sample for detection by gas chromatography.

A collection strategy similar to that used for gas components was employed to collect five halide acid samples for subsequent analysis by ion chromatography. This has allowed the generation of acid concentration profiles for the time interval during and immediately following suppressant addition. The detailed analysis of the agent concentration variation with time in the compartment is given by an accompanying paper.

The different halon replacement agents were evaluated for their effectiveness in extinguishing the fire by comparing their Mean Effective Concentrations and time required to extinguishment. Factors affecting this determination include agent concentrations, nozzle geometry, and validity of the gas sampling and analysis systems.

Quantification of Parameters in the Assessment of Halon 1301 Replacement


Naval Research Laboratory
Navy Technology Center for Safety and Survivability, Code 6180
Washington, D.C. 20375-5320 (202) 767-2254 Fax (202) 767-1716

Introduction

Halons are very effective chemical inhibitors of flaming combustion. They are also potent depleters of the stratospheric ozone layer if they can migrate through the tropopause. We are conducting a halon replacement program that spans from developing an understanding of fire suppression, ozone depletion interactions, intermediate scale evaluation of halon replacement on through full scale evaluation on our fire research ship facility, the ex-USS SHADWELL. This paper discusses the quantification of the operational parameters related to the use of and the effectiveness of a total flooding halon fire suppression system.

Background

The current doctrine for halon 1301 systems specifies that 1) the agent be discharged within 10 seconds, 2) that a homogeneous mixture with a halon 1301 concentration equal to or exceeding the design concentration is achieved throughout the protected space, and 3) that the concentration is maintained for the length of time required to ensure complete extinguishment and for mitigation of persistent ignition sources (1,2).

The halon fire suppression discharge time is defined by the National Fire Protection Association as the time between the cylinder actuation and when the flow from the nozzle becomes predominately vapor (3). This point in time is recognized as the inflection point on the nozzle pressure curve and the point in time when the nozzle temperature drops.

Mixing rates of the considered agent and compartment resident gases are very important in the rapid extinguishment of a fire using a total flooding system. Mixing under these conditions is dominated by the turbulence driven by the rapid injection of agent and the charge gas. The charge gas used by the Navy is nitrogen at a pressure of 4.1 MPag (600 psig) (3).

Model Description

The model used for quantifying the operational parameters in the assessment of halon 1301 replacement is a two symmetrical zone model of a total flooding compartment. The continuity equations for the concentration of a considered agent in each of the two zones may be written:

\[
\frac{dH_L}{dt} = D_L \cdot \frac{1}{\tau_L} H_L \cdot \frac{1}{\tau_C} (H_L - H_U) \tag{1}
\]

\[
\frac{dH_U}{dt} = D_U \cdot \frac{1}{\tau_U} H_U \cdot \frac{1}{\tau_C} (H_L - H_U) \tag{2}
\]
where $H_L$, $H_U$ are the concentration of the halon (lower and upper compartment respectively); $D_L$, $D_U$ are the effective discharge rates into the respective zones. The effective discharge into the lower zone only is considered here and is taken to be a simple exponential function, related to the discharge nozzle pressure decay:

$$D_L = ae^{-bt} \text{ and } D_U = 0$$

where $a$ and $b$ are constants. $\tau_L$, $\tau_U$ are the lower and upper zone agent removal rates (physical and/or chemical), $\tau_C$ is the mixing time between zones and initial conditions are $t=0$, $H_L=0$, $H_U=0$. $H_T$ the total cumulative concentration of agent discharged into the compartment at any given time $t$ is:

$$H_T = \frac{a}{b} (1 - e^{-bt})$$

(3)

The halon 1301 concentration-time profile (4) has a high correlation with the form of equation (3). For sufficiently long times (-bt < c -1) the compartment mixing is readily approximated as:

$$\tau_C = \frac{R^{-1}}{b} \text{ where: } R = \left( \frac{H_L}{H_U} \right)_{t \to \infty}$$

(4)

i.e., $R$ is the lower zone to upper zone concentration ratio at steady state. The discharge time, $t_d$ here is defined quantitatively as $3b^{-1}$.

Efficiency of a Considered Agent

The graphical integration of the derivative curve of the discharge concentration - time profile curve from cylinder activation, $t_o = 0$, to the time of fire extinguishment, $t_e$, is the measure of the system effective extinguishment concentration, $C_e$. The effective extinguishment concentration divided by $t_e$ defined the time average effective application rate of a considered halon agent.

Halide Acid Production

The above simple symmetrical two-zone model coupled with a first-order assumption with respect to the concentration of halon agent present and the rate of halide acid production allowed for a parametric determination of halide acid production rate.

Supported in part by the Naval Sea System Command

References

Theoretical Investigation of C-H and C-X Bond Energy Correlations with OH Reactivity and Fire Suppression Efficiency

Marc R. Nyden
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

and

Karl Irikura
Chemical Sciences and Technology Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

ABSTRACT

The major objective of this investigation is to develop a strategy, based on the calculation of carbon-hydrogen (C-H) and carbon-halogen (C-X) bond dissociation energies, to select the best candidates for the replacement of halon fire extinguishants. In the process, we hope to broaden the scope of agent screening beyond the 11 candidates already identified by the USAF.

The mode of action by which halons extinguish fires has never been fully resolved. The predominant view, however, is that they inhibit chain-branching reactions by promoting recombination of H and OH radicals. The formation of the corresponding hydrogen halide (HX) is thought to be an important step in the chemistry of halon extinction. Once formed, these compounds react with H and/or OH radicals via $HX + H \rightarrow H_2 (H_2O) + X$. An alternate hypothesis, which has also gained wide acceptance, is that halons work simply by absorbing heat from the fire. Once absorbed, this energy is dissipated in vibrational excitation culminating in the formation of HX and other noncombustible reaction products. In either case, the rate of dissociation of the halogen atom (X) from the halocarbon molecule (RX) is critical. Indeed, the empirical observation is that suppression efficiencies increase with decreasing bond dissociation energies in accordance with: R-F < R-Cl < R-Br < R-I.

It is generally conceded that any compound containing Cl, Br or I is a potential threat to stratospheric ozone. Perfluorinated compounds, however, are not thought to pose a problem because HF is chemically and photolytically inert in the troposphere. The principal determinant for ODP is whether the molecule which contains the Cl, Br or I atoms is sufficiently stable to reach the stratosphere. This,
in turn, is largely determined by its propensity to react with OH. Once they escape the troposphere, halocarbons are rapidly photolyzed by high frequency UV radiation in the stratosphere. The halogen atoms released in this process are then free to catalyze the destruction of stratospheric ozone through the following reaction sequence: \( X + O_3 \rightarrow XO + O_2; XO + O \rightarrow O_2 + X \).

The abstraction of H by OH to form \( H_2O \) and the corresponding free radical fragment is the rate limiting step in the chemical processes involved in removing saturated halocarbons from the troposphere. Thus, it is clear that ODP should correlate with C-H bond energies. Similar reasoning suggests that the fire suppression efficiencies of halocarbons should correlate with C-X bond energies. It is possible to make qualitative assessments of these properties on the basis of simple structure/activity relationships. This was, in fact, the strategy employed in constructing the exploratory list of 103 halon alternatives (NIST TN1279) which is the point of departure for the additional agent screening component of the halon project. However, since small differences in bond energies can result in large differences in chemical reactivity, a more quantitative approach will be adopted in the final selection process.

Recent advances in the theory and application of computational molecular quantum mechanics have made the calculation of chemically accurate bond energies (\( \pm 3 \text{ kJ/mol} \)) in two and three carbon halons a realistic goal. Some of the most promising methods, including the G1 and non-local spin density approximations, will be reviewed. Calculations of C-H and C-X bond dissociation energies for a series of halon alternatives are underway. The progress we have made in developing accurate correlations between these bond dissociation energies and measurements of OH reactivity and fire suppression efficiency will be reported, along with a discussion of the strategy we are using to select environmentally safe fire extinguishants.
As part of a program for the U.S. Air Force to identify alternatives to halon 1301 for in-flight aircraft fire protection, the physical behavior of eleven fluorinated compounds upon sudden release from a high pressure cylinder is being investigated. The effectiveness of these agents in suppressing a fire are thought to depend as much on their thermodynamic properties, their behavior during two-phase flow, and the timing of their release, as on their interaction with flame chemistry and the nature of the fire. A series of experiments have been designed to examine the fluid mechanical characteristics separate from the interaction of the agents with the combustion chemistry. Control of those phenomena which dominate the actual suppression process is sought, be they the chemical reactivity of the agent, the thermal quench or dilution provided by the agent, or the properties associated with the physical mixing of the agent into the fire. The adequate protection of dry bay areas of military aircraft requires that an agent be dispersed throughout a one cubic meter volume within 50 ms of an ignition event. The fire suppressant storage vessel is typically adjacent to the space being protected. The issue of concern in the current study is how quickly the material can exit the vessel and be dispersed throughout the volume at concentrations sufficient to extinguish an accelerating turbulent flame.

A high pressure vessel equipped with a quick release opening and an internal volume of 500 ml has been built in order to assess the discharge dynamics of the eleven agents mixed with nitrogen. A rupture disc 18 mm in diameter, mounted at the opening of the vessel, is used as a quick release mechanism for discharging the contents of the vessel. The effect of the size of the opening on the nature of the emptying process will be assessed by placing various diameter orifices (from 6 mm to 18 mm) at the exit, with the constraint that the emptying time be less than 500 ms.

The vessel is initially charged with a predetermined amount of agent and subsequently pressurized with nitrogen to 4.1 Mpa. The mass of agent is controlled so as not to exceed the safe operating pressure of the vessel (8.2 Mpa). The pressure rating of the rupture disc is chosen to be within 5% of the value predicted from static pressure measurements at temperatures between 213 K and 422 K. During a test, the charged vessel is placed in a constant temperature bath until thermal equilibrium has been reached. The contents are discharged vertically upwards by impulsively injecting a small amount of nitrogen into the vessel, causing the disc to rupture. The internal temperature and pressure of the vessel are monitored during discharge. The tests are documented using high-speed cinematography.

The fluid flow is multi-phase and choked as it passes through the exit orifice. The agent’s concentration in time and space outside of the vessel is strongly coupled to its thermodynamic and transport properties, and the turbulent entrainment of the surrounding ambient air. The near field is composed of a dense two-phase spray, which rapidly evaporates to form a free turbulent jet in the far field. The boundary of the two-phase region is determined by laser sheet illumination and schlieren photography. The schlieren photographs provide information about both the droplet loading and the gas density. The laser sheet illumination with digital image processing can provide estimates of the droplet velocities. High speed photography yields temporal information on droplet dispersion and vaporization. It is planned to use laser-line illumination with a linear diode array, scanning at 2.6 kHz, to make a small number of vertical measurements of the Mie scattering intensity. Rayleigh scattering will be used to characterize the gas-phase mixing. The gas phase experiments will include line and 2 d imaging.
In parallel with the jet penetration experiments, the numerical code KIVA-II has been adapted to simulate the transient multi-phase flows typical of the agent release problem. The code is limited to the domain downstream of the exit orifice, and is not expected to yield quantitative predictions because the initial state of the exiting spray is highly uncertain; however, the code is useful for parametric studies of the agents and for the assessment of the significance of such effects as cross flow and orientation on agent dispersion.

<table>
<thead>
<tr>
<th>Chemicals to be Examined: CH₂F₂, CH₂F²/CHF₂CF₃, C₃HF₇, CHF₂Cl, CH₂FCF₃, C₂F₆, CHFCICF₃, CHF₂CF₃, C₂F₆, C₄F₁₀, cyclo-C₄F₈</th>
</tr>
</thead>
<tbody>
<tr>
<td>Charging gases: nitrogen, CHF₃</td>
</tr>
<tr>
<td>Initial charging conditions: 4.1 MPa, 295 K, variable mass</td>
</tr>
<tr>
<td>Pressure vessel volume: 100 ml (static), 500 ml (dynamic)</td>
</tr>
<tr>
<td>Maximum design pressure: 13.7 MPa (static), 8.2 MPa (dynamic)</td>
</tr>
<tr>
<td>Test conditions: 213 K, 295 K, and 422 K</td>
</tr>
<tr>
<td>Pressures: equilibrium at test temperatures</td>
</tr>
<tr>
<td>Vessel release mechanism: pressure actuated rupture disc, 6 mm to 18 mm diameter</td>
</tr>
<tr>
<td>Vessel orientation: upward, downward, horizontal (with and without extension)</td>
</tr>
<tr>
<td>Dependent variables: pressure and temperature as function of time (dynamic tests), pressure as function of temperature and mass (static tests)</td>
</tr>
<tr>
<td>Flow characterization of representative agent: radial and axial boundary of two-phase region, radial and axial boundary of minimum concentration for suppression (as functions of time and initial temperature), bluff body and orientation effects on two-phase and suppression boundaries</td>
</tr>
<tr>
<td>Tests of other agents: centerline two-phase boundary and centerline concentration at one location downstream after release</td>
</tr>
<tr>
<td>Modeling conditions: two-phase transient jet flow with vaporization, gas jet in cross flow, gas-powder jet</td>
</tr>
</tbody>
</table>
CHEMICAL AND PHYSICAL EFFECTS OF WATER DURING FIRE SUPPRESSION

ARVIND ATREYA, TODD CROMPTON AND SANJAY AGRAWAL

Combustion Laboratory
Department of Mechanical Engineering
Michigan State University
East Lansing, MI 48823

(Supported by NIST)

Suppression of fire results from the chemical and/or physical effect of the suppression agent on the burning process. Physical suppression mechanisms act by either cooling the condensed-phase to inhibit fuel pyrolysis (e.g. water application) or by quenching the gas-phase combustion reactions? The gas-phase quenching may be accomplished by simply blowing-off the flame (i.e. by increasing the strain rate) or by diluting the fuel and/or the oxidizer streams with inert gases such as \(\text{H}_2\text{O}, \text{CO}_2,\) and \(\text{N}_2\) (effect of heat capacity, heat of vaporization or heat of decomposition) or by separating the reactants by mechanical or other means (blanketing). Clearly, combined extinguishment actions are also possible and in-fact desirable. These may occur naturally, as in the case of water application, where in addition to condensed-phase cooling, the pyrolyzed fuel is also diluted by evaporated water vapor. The chemical suppression mechanisms, on the other hand, are believed to interfere with the critical reaction steps responsible for maintaining the flame.

Water, the most widely used suppression agent, is thought to be chemically inert in a fire and is believed to have a physical effect. However, there is little quantitative information regarding these effects during extinguishment by water. This work is an attempt to develop such a quantitative understanding of the chemical and physical mechanisms responsible for fire suppression by water. The results of an experimental investigation into extinguishment of a stagnation-point flow diffusion flame supported by a PMMA sample and a methane counterflow diffusion flame diluted by water vapor are presented. While other suppression agents will also be examined, initially this work focuses on water to establish a standard for comparison of suppression effectiveness.

Typical experimental results of the overall species composition measurements made in the exhaust duct of the stagnation-point flow apparatus during suppression by water are shown in the enclosed figure. This figure shows an increase in the \(\text{CO}_2\) production rate and the \(\text{O}_2\) depletion rate upon repeated 0.5ml water application (marked by arrows). This implies that the burning rate first substantially increases before it eventually decreases as a result of water application. For droplets less than 0.3ml, the subsequent decrease was found to be negligible. The increase in the burning rate was substantial and unexpected because it implies that instead of suppressing the flame, it is enhanced by applying water droplets. This increase is also consistent with a decrease in \(\text{CO}\) and total hydrocarbon concentrations and visibly corresponds to the disappearance of soot. Thus, as a result of water evaporation \(\text{CO}_2\), total hydrocarbons and soot are oxidized to \(\text{CO}_2\), depleting \(\text{O}_2\) in the process. This chemical hypothesis is supported by the fact that a non-sooty flame containing little \(\text{CO}\), soot \& unburned hydrocarbons does not show a similar increase in the burning rate due to water application. It seems that there are two simultaneous effects as a result of water application: (i) chemical enhancement of the burning rate (which is important only when the flames are sooty; Note: most fires are sooty), and (ii) physical cooling of the
solid via water evaporation.

The results obtained in the stagnation-point diffusion flame apparatus are consistent with the experiments conducted in sooty (fuel-rich) methane counterflow diffusion flames. Here detailed flame structure measurements were made. It was found that by reducing the \( \text{O}_2 \) concentration while maintaining the flame temperature by preheating the reactants (thus reducing the \( H_2O \) concentration in the reaction zone) led to an early soot inception and increased soot volume fraction. However, direct addition of only 3.6% \( H_2O \) (while holding all other conditions constant) resulted in delayed soot nucleation and a significant reduction in the soot volume fraction. These observations can be consistently explained by the mechanism of \( OH \) interference with soot inception. An increase in the \( H_2O \) concentration (brought about either by an increase in the \( \text{O}_2 \) concentration or by direct addition) results in an increase in the \( OH \) concentration. This reduces the PAH and \( C_2H_2 \) concentrations (the corresponding reduction in total hydrocarbons and CO was observed in the stagnation-point flow diffusion flame over PMMA) and delays soot inception. This substantially decreases the ultimate soot loading and increases the combustion efficiency and hence the burning rate.

REFERENCES


Multi-Droplet Evaporative Cooling:
Experimental Results

H. Dawson, M. di Marzo
Mechanical Engineering Department
University of Maryland at College Park

The phenomena associated with extinguishment of a solid fuel fire are investigated experimentally. In particular, the evaporative cooling of a hot non metallic surface subjected to the impingement of a random array of droplets is quantified. The relevance of this study to the extinguishment process is two-fold: a) it describes the cooling of the surfaces exposed to fire and b) it quantifies the vapor generation in the fire environment.

The experimental results reported here refer to a simplified situation with respect to the actual process to provide a controlled, repeatable and quantifiable set of data which will be used to validate the multi-droplet model which was obtained earlier (Tartarin et al., 1992). The fire environment is simulated by three, temperature controlled, radiant panels mounted above the surface of a macor square tile (0.1524 x 0.1524 x 0.0254 m). The tile is pasted onto a chilled plate which is kept at near ambient temperature. The heaters are controlled on a temperature feedback and are held at a constant temperature set point. These thermal boundary conditions are designed to obtain a linear temperature profile in the tile depth and to insure that the tile exposed surface is isothermal over the spray impingement region, prior to the initiation of the transient.

The droplet distribution used in the experiment is random. However, all droplets are identical and the spray impingement region is kept constant. The water mass flux is controlled by changing the droplet generation frequency. The droplet dispenser is constituted of a cavity which gradually tapers to a small opening at the bottom where a needle pointing downward is attached. The cavity is bounded at its top by a plastic disk which is pulsed downward by a plunger at the desired frequency. The dispenser is suspended from four parallel vertical wires and is moving above the deposition region under the action of three pulsating bumpers which are moving toward each other symmetrically. To insure continuous motion of the dispenser, a periodic displacement of one of the four suspending wires is provided by a rotating device. Figure 1 shows a typical droplet distribution as recorded by a camera looking at the dispenser from above.

The water used in the experiment is de-ionized and de-gassed. In order to obtain a constant feed of water, a mineral oil sealed reservoir and head control tank has been used. This system allows to control the fluid head above the distributor within 0.01 m and at the same time it keeps any gasses from diffusing back into the water. Experiments with de-ionized water are also being conducted to assess the effect of gasses coming out of solution during the evaporative process. Additional consideration will be given to the effect of impact momentum on the droplet shape on the surface, which greatly affects the evaporation rate.

The macor tile transient surface temperature distribution is recorded with an infrared camera. The infrared image is correlated to a temperature distribution (see Klassen et al., 1992 and di Marzo et al., 1992). The 256 shades of gray are related to a temperature scale of about 120 °C. The scale is set by simultaneously measuring the temperature of a given surface point with a thermocouple probe and via the infrared camera reading. The spacial resolution is of about 0.0001 m/pixel and the temperature resolution is of 2 °C/gray-shade which yields an accuracy of about ± 1 °C. The
transient temperature distribution is recorded on a VCR and selected frames during the transient are grabbed by a PC to be analyzed. The camera has a field of view of a circular area with a diameter of about 0.05 m. This area is viewed through a blackpainted chilled pipe which eliminates stray radiation from other sources than the macor surface in direct view. Figure 2 shows a typical snap-shot of the temperature distribution of the sprayed surface. In order to condense the information available, the averaged transient temperature is calculated for selected frames (e.g. Fig. 2) during the evaporative transient. A detailed description of the components of the experimental set-up will be provided along with a sample data record of the phenomena to illustrate the data processing. The surface averaged transient temperature for various conditions will be presented and qualitatively compared with preliminary results of the numerical simulation (Tartarini et al., 1992).

Acknowledgements: The authors are indebted to Dr. Evans (BFRL-NIST) for his guidance and to the Buildings and Fire Research Laboratory for the support of this research program.

References


MODELING OF TURBULENT FLAME RADIATION

by

J. de Ris and L. Orloff
Factory Mutual Research Corporation

Radiation from turbulent diffusion flames controls the burning rates of moderate and large-scale fires and is therefore of central importance to the flammability problem. Recently we have developed a global flame radiation model which predicts the radiant fraction from buoyant turbulent (fuel-jet) diffusion flames in terms of the fuel's laminar flame smoke-point value. The model considers the separate roles of soot and gaseous radiation. The effective flame radiation temperature is predicted by the model in terms of the: 1) incompleteness of combustion, 2) radiant heat loss and 3) turbulent mixing with good agreement with experiment.

It is argued that the soot absorption coefficient (or soot volume fraction) is: 1) proportional to the turbulent macro-scale flow time in the soot formation region of the flames, and 2) inversely proportional to the chemical time for soot formation. The chemical time for soot formation is shown to be approximately proportional to the smokiness of fuel and/or oxidant mixtures show that: 1) the soot formation is quite sensitive to the local velocity of the gas flowing through the diffusion flame toward the fuel side, while 2) the soot formation is insensitive to the adiabatic stoichiometric flame temperature, T_{ad}.

The predicted radiant fractions are in excellent agreement with experiment for different oxidant and fuel mixtures covering a wide range of flame sootiness. The model is tested against both axisymmetric and planar free-jet buoyant turbulent diffusion flames over a range of heat release rates. We are currently applying the model to wall fire flames which have smaller radiant fractions due to: 1) convective heat loss to the surface, 2) significant dilution of the supplied fuel gases by products of combustion diffusing back toward the fuel source, and possibly 3) reduced macro-scale mixing times due to wall shear stress.

The model requires knowledge of the fuel's chemical structure. Methods are now available for evaluating the required smoke-point of gaseous, liquid, and solid (charring and non-charring) fuels of interest.

By developing algebraic curve-fit formulas for the soot and gaseous emissivities, the model is self-contained and can be readily implemented using spreadsheet software.

REFERENCES


2. de Ris, J. and Orloff, L.: submitted to Combustion and Flame.


The scientific understanding of wall-fir combustion and in particular the development of pertinent theoretical models require experimental data on flame radiation so that these models can be applied to fires of hazardous scale where radiative energy transfer plays a controlling role. The purpose of the present work is to extend the range of such data beyond the upper limits of fuel sooting tendency and flame height attained in previous investigations. In the approach used here, wall fires of solid fuels are simulated by burning gaseous hydrocarbon fuels under steady-state conditions. The four fuels, methane, ethane, ethylene and propylene, are used to cover a sufficiently wide range of sooting tendency.

Initially, the study concentrated on the simulated overfire region by releasing the entire fuel flow from a horizontal slot burner placed adjacent to a vertical water-cooled metal plate while more recent work is concerned primarily with the pyrolysis region, simulated by supplying gaseous fuels from a water-cooled vertical sintered-porous-metal surface. The burner developed for this study consists of ten 132-mm-high and 380-mm-wide vertical porous-metal panels topped by a 660 mm high water-cooled solid-metal heat transfer plate divided into five similar segments. To approach two-dimensional flow conditions as closely as possible, water-cooled side walls of 150 mm depth are attached to the burner over the entire height of 1980 mm.

The water cooling passages embedded in the porous metal and the heat transfer plate are instrumented with differential thermocouples for measurement of the total heat transferred to each panel and segment by the flame. Other instrumentation includes a wide-view-angle radiometer measuring total radiant emission and a slit scanning radiometer to determine the vertical distribution of radiant emission by horizontal slices across the wall fire.

In the slot-burner work it was found that the radiative fraction of total heat release rate was reduced significantly by placing the burner adjacent to a water-cooled wall, compared to values obtained with free-burning slot-burner or jet flames. This work also yielded dimensionless correlations of radiant emission as a fraction of height independent of heat release rate, by introducing a radiation flame-length parameter. The latter varied with the 1/2 power of heat-release rate, suggesting that flame heights are controlled primarily by radiant cooling rather than turbulent mixing, and implying a mean volumetric heat-release rate independent of fire size and fuel type.

Work with the porous-metal burner showed that for any fixed fuel mass flux, the radiance \( N_\phi \) increases linearly with height, \( z \), after an initial jump at the flame base. At fixed \( z \), linear relationships between reciprocal radiance \( N_\phi^{-1} \) and reciprocal mass flux \( m^{-1} \) were obtained. For methane and ethane these relationships held over the entire range of mass flux, while for ethylene and propylene, an abrupt transition separating two regimes occurred as critical mass fluxes. The ordinate

*Supported partly by the Building and Fire Research Laboratory, National Institute of Standards and Technology, and partly by Factory Mutual Research Corporation.
intercept $\eta$ representing an asymptotic radiance, and the slope $s$ of the linear relationship, are functions of height $z$. $\eta$ and $s$ increase with fuel sooting tendency at fixed $z$.

Heat fluxes to the porous metal panels decreased with increasing mass flux near the flame base, but were nearly independent of mass flux near the top portion of the porous-metal burner. These heat fluxes increased with fuel sooting tendency, except for a reversed trend between methane and ethane. At sufficiently high mass flux, a jump of heat flux occurred at the junction between the porous-metal burner and the solid-metal heat transfer plate, and the heat flux increased further with height in this over-fire region. In this zone, the data were separated into a lower overlapping set for alkane fuels, methane and ethane, and a higher overlapping set for the alkanes, ethylene and propylene.

Additional results beyond those included in Ref. 11 will be presented. In particular, separation of total heat flux to the wall into its radiative and convective components is an essential requirement for the development of a predictive theoretical model. Instrumentation for performing pertinent measurements is currently being developed. However, preliminary estimates of the separation can be obtained from the current data and will be reported.

REFERENCES

HEAT FEEDBACK TO A POOL FIRE

A. Hamins, M. Klassen', J. Gore', and T. Kashiwagi
National Institute of Standards and Technology

The heat flux to the surface of a pool fire and the mass flux vaporizing from the pool are part of a positive feedback loop. The rate of mass burning depends on the heat feedback from the flame to the fuel surface and the mass burning rate controls the total heat release and the amount of heat feedback. Heat feedback, \( Q \) (W), to the pool surface is due to the sum of conduction (\( Q_{\text{cond}} \)), convection (\( Q_{\text{conv}} \)), and radiation (\( Q_{\text{rad}} \)). An energy balance at the fuel surface of a steady-state fire requires that the heat feedback is equal to the energy needed to vaporize the fuel and to overcome heat losses via reflection and re-radiation by the fuel surface to the surroundings (\( Q_{\text{rerad}} \)), increased sensible heat of the liquid pool (\( Q_s \)), and heat flux to the bottom and sides of the pool burner (\( Q_{\text{loss}} \)):

\[
Q = Q_{\text{cond}} + Q_{\text{conv}} + Q_{\text{rad}} = M \cdot (H_v + C_p \cdot (T_s - T_a)) + Q_{\text{rerad}} + Q_s + Q_{\text{loss}}
\]

where \( M \) (g/s) is the mass burning rate, \( H_v \) (J/g) is the heat of vaporization at \( T_s \) (K), \( T_s \) (K) is the pool surface temperature, \( C_p \) (J/g-K) is the fuel heat capacity, and \( T_a \) (K) is the ambient temperature. The loss terms (\( Q_{\text{rerad}}, Q_s \), and \( Q_{\text{loss}} \)) diminish the fraction of energy available for fuel vaporization.

Although much research has been conducted on the non-homogeneous properties of pool fires, most work has assumed a uniform mass burning rate across the entire pool surface, independent of radial location \( r \). Only a few experiments have measured localized heat transfer at the pool surface. In this study, \( Q_{\text{rerad}} \), \( Q_s \), and \( Q_{\text{loss}} \) were independently measured, \( Q_{\text{rerad}} \) was estimated and \( Q_{\text{conv}} \) was calculated using Eqs. 1 and 2 (\( Q_{\text{cond}} \) measured to be negligible).

A steel fire pan composed of 4 concentric rings of 0.077, 0.152, 0.229, and 0.303 m in diameter, water cooled at the bottom, was used to measure the radial distribution of burning rate. The fuel flow to each ring was gravity fed and independently controlled such that the liquid level was maintained 5 mm below the rim. The fuel for each ring was stored in separate reservoirs, each on a load cell (0.1 g repeatability), such that the mass loss rates could be monitored. A typical experiment consisted of 10 minutes of warm-up and 45 minutes of observation. Experiments were repeated several times. Heat losses (\( Q_{\text{loss}} \) and \( Q_s \)) were monitored by Ch/Al thermocouples. The fuels tested were methanol (no soot), heptane (moderately sooting) and toluene (heavily sooting). Figure 1 shows the burning rate of methanol, heptane, and toluene in the annular ring pool. The burning rate profile is relatively flat for all fuels.

A narrow view angle (0.004 ster or ±7°), nitrogen purged, water cooled radiometer was used to measure the radiative flux at the pool surface. The radiometer face was positioned 2 mm above the fuel surface. Measurements along the pool surface were made at 2 cm intervals and tilted at 20° intervals from 30° to 150° (normal to pool surface = 90°).

*supported by NIST Grant 60NANB09D0944. Permanent address: School of Mechanical Engineering, Purdue University, West Lafayette, IN 47907
Data was taken at a 30 Hz sampling rate and averaged for 90 s. The effect of gauge heat up was measured with the fire burning, by placing a water cooled cap over the radiometer (nitrogen purge maintained). The radiative intensity at each angle was numerically integrated to yield the total radiative flux at a particular location, The radiometer was calibrated using a standard flux gauge. For all fuels and all locations, the largest intensity incident on the radiometer was at $\theta = 90^\circ$. The figure below shows that the integrated radiative flux was highest for toluene, followed by heptane and then methanol. The flux decreases from the pool center towards the pool rim for methanol, but was relatively flat for heptane and toluene.

The measured values of the sum of $Q_s$ and $Q_{loss}$ amounted to 1%, 5%, and 8% of the energy required for fuel vaporization, $[\bar{M} \cdot (\bar{H}_u + C_p \cdot (T_s - T_a))]$, for the methanol, heptane and toluene fires respectively. These values and the local mass burning rate data were used to calculate $Q_{conv}$. A comparison of the local radiative and convective heat flux shows that at pool center, for all fuels, the majority of the heat flux is due to radiation. Convective transfer plays a major role only in the methanol fire, where near the pool edge about half of the heat flux is due to convection and half to radiation. At the pool center, convection accounts for only a small amount of the heat transfer in the methanol fire, In the heptane fire, radiative transfer dominates throughout the pool. Near the pool center radiation accounts for almost all of the heat transfer, while near the pool edge convection plays a small role. In toluene, heat transfer over the entire surface is dominated by radiation.

---

Integrated radiative heat flux as a function of radial location in 0.30 m pool fires.
THE ROLE OF REFLECTION AND FUEL VAPOR ABSORPTION IN RADIATIVE HEAT FEEDBACK TO POOL FLAMES

M. Klassen, Y.R. Sivathanu and J. P. Gore
School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907

Introduction
Radiation from liquid fueled flames is an important heat transfer mechanism in combustion systems. Radiative heat flux from fires surrounding objects determines the possibility of ignition, flame spread and flashover. In liquid pool flames, the energy required for fuel vaporization is received entirely from the flame.

Measurements of the incident radiative heat flux at the fuel surface in 30 cm toluene pool fires were found to be approximately 35 kW/m² across the radius of the pool [1]. Measurements of the feedback based on local burning rate indicate that only 31 kW/m² are used. A similar discrepancy was found by Tewarson and Pion [2]. Recent measurements in 60 cm and 1 m pool fires show an even larger discrepancy between measured incident radiation and heat flux required to support the burning rate. Important factors which may explain this discrepancy include reflection of radiation off the fuel surface and the absorption of radiation by fuel vapor.

Experimental Methods
Measurements of the incident radiative heat flux at the fuel surface were made using a narrow-angle radiometer. Simultaneous measurements of the emission/absorption properties of flames were made using a three wavelength optical pyrometer [1]. Measurements of the fuel surface infrared reflection were made using a blackbody source and lead sulfide (PbS) detector (Fig. 1). Toluene vapor absorption measurements were made in-situ in regions of no measurable emission.

Results and Discussion

Fig. 2 (A) shows the transmittance of fuel vapor over a 8 mm path. A large dip occurs between approximately 3200–3600 nm. But little flame emission is found at the corresponding wavelengths (Fig. 2 (B)) . This indicates that fuel vapor absorption plays a minor role in radiative transfer in toluene flames.

Fig. 3 presents measurements of the infrared reflectivity of several different liquids. Large reflectivity is found at small incidence angles for all fuels. However even in toluene flames where low regions (x/D <1.0) of the flame dominate feedback [3] reflection proves to play only a minor role in the surface energy balance. Fig. 4 shows the effect that reflection has on predictions of radiative heat flux.

References

Acknowledgements
The authors acknowledge the contributions of Dr. A. Hamins and Dr. T. Kashiwagi of NIST. This work was supported by NIST Grant Nos. 60NANB09D0944 and 60NANB1D1169.
**Fig. 1** Fuel surface reflection measurement set-up

**Fig. 2** (A) Toluene fuel vapor transmittance
(B) Toluene flame emission

**Fig. 3** Fuel surface reflection for various liquids

**Fig. 4** Predictions of incident radiative heat flux with and without reflection
Total Radiative Heat Loss in Jet Flames from Single Point Radiance Measurement

P. Dutta, Y. R. Sivathanu, J. P. Gore
School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907

Introduction: The total energy radiated by fires ($Q_r$) to the surroundings is of importance in determining burning rates, flame spread and evaluating safety standards for buildings and materials. The total radiative fraction, $X_R$, (fraction of chemical energy radiated to the surroundings) is a characteristic of the fuel over a range of heat release rates, $Q_r$. The total radiative heat loss are usually estimated either by measurements of radiance over a semi-infinite enclosure surrounding the flames [1-2] or from a single point measurement assuming spherical isotropy [3]. For liquid pool flames, a single point radiance measurement can provide reasonable estimates within 13% of total radiative heat loss from the flames [4]. The objectives of the present work are (1) to determine the efficacy of a single point technique for both single-phase and multi-phase jet flames experimentally and (2) to study the theoretical basis for the estimation of total radiative flux from single point radiance measurements. The results would be useful in estimating the radiative output of large scale oil-well fires and flares.

Theoretical and experimental methods: The scalar property distributions essential for radiation calculations were obtained from a k-e-g model coupled with the laminar flamelet concept for soot volume fractions and temperatures. The radiative heat flux ($q_p$) to different representative detector locations were calculated using a multi-ray approach [5]. The radiative flux were normalized using dimensional analysis and the results plotted in these normalized coordinates to seek universal curves. The experimental work consisted of making radiance measurements along a semi-infinite enclosure surrounding the flames. The resulting radiance measurements are plotted in the same normalized coordinates. The single phase jet diffusion flames considered were ethylene/air or methane/air stabilized on a 6 mm diameter burner. Multi-phase jet flames burnt a mixture of methane and methanol, heptane or crude oil in an effervescent atomized burner. During previous tests of crude oil/methane flames from a pressure atomized burner, more than 50% of the fuel was unburned due to poor atomization quality. In contrast to the above, the effervescent atomized burner resulted in complete combustion of the crude oil for relatively low gas to liquid mass ratios (GLR). Visual flame heights were obtained from high speed photographs.

Results and Discussion: Figure 1 shows the calculated heat flux, at a radial distance $R$ and an axial distance $x$ from the burner exit, plotted as a function of axial position in the normalized coordinates at half the stoichiometric flame height (0.5 $L_f$), 1.0 $L_f$, and 2.0 $L_f$. The normalized heat flux, for the six different conditions shown in Fig. 1., collapse to approximately a single curve showing the similarity of radiative fluxes from the turbulent jet flames. Figure 2 shows the experimental data from radiance measurements at half the visible flame height ($L_f$) for two different flow conditions for methane/air and ethylene/air diffusion flames. The experimental data for the two different fuels, with varying radiative heat loss fractions, collapse onto a single curve supporting the choice of scaling parameters. Figure 3 shows the experimental heat flux at a radial distance equal to the visible flame height from two-phase diffusion flames plotted on the normalized coordinates. Similar to the single phase diffusion flames, the normalized heat flux profiles for the two phase jets collapse onto a single curve. For all the crude-oil test conditions, the radiative fractions depended on the gas to liquid ratio (GLR). For fixed GLR, the radiative fraction was found to be independent of fuel flow rate. The radiative fraction decreases with increasing GLR due to the lower sooting tendencies of the methane flames. In spite of the different radiative fractions and the different GLRs, the normalized heat flux profiles collapse. Based on these findings, a single radiance measurement at a given $R/L_f$ and $x/L_f$ can be used to estimate the total radiative heat loss from single phase and two phase jet diffusion flames using Figures 2 and 3 respectively.
Acknowledgments. This study is supported by the National Institute of Science and Technology under Grant Nos. 60NANB9DO834 and 60NANB1D1172 with Dr. D. D. Evans Serving as NIST Technical Officer.

References:

Fig. 1. Normalized heat flux profiles along the axis for three different radial locations from multi-ray calculations.

Fig. 2. Normalized heat flux profiles along the axis of single phase diffusion flames.

Fig. 3. Normalized heat flux profiles along the axis of two phase diffusion flames.
Specific Absorption Coefficients of Soot Particles
in Laminar Flat Flames

Y. R. Sivathanu and J. P. Gore
School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907

Introductions. Theoretical predictions of continuum radiation from luminous flames require information on volume fractions, temperatures and specific absorption coefficients of soot particles. Past studies [1-3] show that the specific absorption coefficients of soot particles are independent of fuel type (C/H ratio) and weakly dependent on temperatures. In contrast to the above, two recent studies [4,5] show substantial C/H ratio effects on the specific absorption coefficients in the infrared region making heat transfer calculations in strongly radiating flames intractable. These studies [4,5] involve a single operating condition for which the optical homogeneity of the path was not considered. The specific objectives of this study were: 1) to measure the specific absorption coefficients of soot particles for a wide variety of conditions for three different fuels, and 2) study the effect of optical inhomogeneity on the measurements.

Experimental methods. Premixed methane/oxygen, propane/oxygen and ethylene/oxygen flames were stabilized by a ceramic disc placed 50 mm above the surface of McKenna flat flame burner of 5 cm diameter. A four-wavelength emission/absorption probe shown in Fig. 1. was used to measure the absorption soot volume fractions (from the extinction of a He-Ne laser at 632 nm) and the emission soot volume fractions and temperatures (from emission intensities at 800 nm and 900 nm) at a height of 15 mm above the surface of the burner. Equality between emission and absorption soot volume fractions is taken as an indicator of optical homogeneity and for these conditions, the emission intensities measured at 2300 nm, and 4000 nm were used to infer the specific absorption coefficients at these wavelengths. A range of equivalence ratios and flow rates were used to change the residence times and temperatures of the soot particles in the flame.

Results and Discussion. Figure 2 shows the specific absorption coefficients obtained using the refractive indices given by three investigators [1,2,5] as a function of wavelength. The present experimental data averaged over all optically homogeneous conditions are also shown for comparison. The results show that the specific absorption coefficients in the infrared region are independent of fuel type (C/H ratio) even when fuel dependent refractive indices were used in the visible region. Figure 3 shows the specific absorption coefficients plotted as a function of optical path homogeneity defined as the ratio of emission to absorption soot volume fraction. Measurements outside the optically homogeneous band show a factor of 3 departure from the correct values. Hence, if different optically homogeneous conditions were used for different fuels, the results would show a spurious fuel-type effect. Figure 4 shows the radial profiles of temperature and soot volume fraction for two different equivalence ratios calculated using a PDE solver. Entrainment of shroud gas due to buoyancy is the major cause of optical inhomogeneity. The degree of inhomogeneity and the inferred specific absorption coefficients at 4000 nm are also tabulated. Higher temperatures result in greater inhomogeneity causing spuriously high value for the inferred specific absorption coefficients.

Acknowledgements. Partial support for this study is provided by the National Science Foundation Grant Number CTS-8914520.

References:
Fig. 1. Sketch of four line absorption/emission probe.

Fig. 2. Comparison of the specific absorption coefficients from different models with the present data.

Fig. 3. Specific absorption coefficients for $\lambda = 4000$ nm as a function of degree of inhomogeneity.

Fig. 4. Calculated radial profiles of soot volume fractions and temperatures.
Scattering by Soot Aggregates in Turbulent Diffusion Flames

R. Mital, J. Dylla, Y. R. Sivathanu, J. P. Gore
School of Mechanical Engineering
Purdue University
West Lafayette, IN 47907

Introduction. Radiation heat transfer studies in luminous flames have assumed negligible effect of scattering by soot particles due to their relatively small diameters compared to the wavelengths of interest [1-3]. Recent morphological studies of soot aggregates in diffusion flames show that the scattering coefficients of soot particles may be as high as 100% [4-5]. However, these experiments were conducted either in laminar flames [4] or in the overfire region of turbulent flames [5]. In addition, the effect of scattering on radiative heat transfer has not been studied. Based on these observations, the objectives of the present study are: 1) to measure the scattering coefficients in turbulent diffusion flames and 2) to evaluate the effect of the measured coefficients on radiative heat transfer.

Theoretical and Experimental methods. In situ measurements of the scattering coefficients were conducted for a turbulent acetylene/air jet diffusion flame. The exit Reynolds number of the flame was 9200 based on a burner diameter of 6 mm. Fig. 1 shows a sketch of the present scattering/absorption probe, which consists of two fixed and one movable stainless steel purged tubes. The extinction of a He-Ne laser was measured across a probe length of 10 mm and the intensity of the scattered light was measured at three different angles (450, 900, and 1350) providing an estimate of the instantaneous scattering coefficient. The measured scattering coefficients were used in multi-ray radiative flux calculations [3] to estimate the effect of scattering on the radiative transfer from a representative horizontal path in the flame.

Results and Discussion. Table 1 shows representative value of the scattering coefficients at an axial location corresponding to x/d = 50. The mean value of the scattering coefficient is the highest at 450 (forward direction). The fluctuations are relatively low at large angles but increases to about 77% of the mean value at 450. Mean soot volume fractions and temperatures obtained from a coupled k-e-g calculation [3] along with a mean scattering coefficient of 0.33 were used to evaluate the effect of scattering on the spectral radiation intensity (at 900 nm) along a diametric path in the flame. Fig. 2 shows the distribution of the different terms in the equation of radiative transfer along the diametric path. The emission intensity is symmetric since the scalar distribution obtained from the calculation is symmetric. The intensity absorbed along the path depends on the incident intensity and is comparable in magnitude to the local emission. The outscatter, similar to the absorption depends on the incident intensity along the path and is as high as 25% of the local emission. The inscatter of radiation intensity depends on the incident intensity from all parts of the flames and is therefore symmetric. Fig. 3 shows the net radiation intensity leaving a local segment as a function of the distance along the segment. This represents the balance between the different terms plotted in Fig. 2. At different points along the segment, the effects of neglecting scattering can be as high as 30%. However, the inscatter of radiation intensity into the segment tends to cancel the outscatter of radiation intensity from the segment. The difference between the values of intensity leaving the entire segment neglecting scattering and those including scattering is only about 5%.

Acknowledgements. This study is supported by the National Science Foundation under Grant No. CTS-9157920.

References:
Figure 1: Sketch of scattering / absorption probe.

Figure 2: Distribution of the terms in the equation of transfer along the representative radiation path.

Figure 3: Radiation intensity as a function of distance along the representative radiation path.

Table 1: Scattering Coefficients in Acetylene/Air Flames, $Re = 9200: x/d = 50; r/x = 0.0$

<table>
<thead>
<tr>
<th>Angle</th>
<th>45</th>
<th>90</th>
<th>135</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>0.33</td>
<td>0.11</td>
<td>0.09</td>
</tr>
<tr>
<td>RMS</td>
<td>0.77</td>
<td>0.20</td>
<td>0.22</td>
</tr>
</tbody>
</table>
STRUCTURE AND OPTICAL PROPERTIES OF OVERFIRE SOOT IN BUOYANT TURBULENT DIFFUSION FLAMES

by

U. O. Köylü and G. M. Faeth
Department of Aerospace Engineering
The University of Michigan
Ann Arbor, Michigan 48109-2140

The physical and optical properties of soot were measured within the fuel-lean (overfire) region of buoyant turbulent diffusion flames in still air. The study was limited to pool-like fires in the long residence time regime, where soot and carbon monoxide emission factors are independent of flame size and position in the overfire region. Both gas and liquid fuels (acetylene, propylene, ethylene, propane, toluene, benzene, n-heptane, and isopropanol) were studied to provide information for a range of H/C ratios (1-2.67) and fuel types. Measurements included thermophoretic sampling and TEM to find soot structure, ultimate analysis to find soot composition, and laser scattering and extinction to find soot optical properties, see \([4-6]\) for more details.

Soot aggregate properties varied with the fuel but not with position or residence time in the long residence time regime. Primary particle diameters were 30-51 nm with average numbers of primary particles per aggregate of 255-552 — both increasing with increasing propensity of the fuel to soot. Aggregate size distributions were unusually broad (typically 30-1800 primary particles per aggregate) but could be represented reasonably well using a log-normal function. The aggregates were mass fractal-like, with fractal dimensions of 1.7-1.8. Elemental mole ratios of soot composition were variable, suggesting potential for variable refractive indices, e.g., \(C/H = 8-18\), \(C/O = 58-109\) and \(C/N = 242-976\).

Soot structure and laser scattering measurements were used to develop predictions for soot optical properties. These predictions were based on existing Rayleigh-Debye-Gans (RDG) fractal aggregate optical theories, extended to consider the broad size distributions of overfire aggregates. Predicted and measured angular scattering patterns (scattering cross sections for various polarization states) are illustrated in Fig. 1 for acetylene soot at a wavelength of 514.5 nm. The results indicate large departures from the small particle (Rayleigh) scattering approximation (e.g., forward scattering is 100-1000 times larger than wide angle scattering), with excellent agreement between RDG predictions and measurements over the accessible range of scattering angles. The performance of the theory for soot from other fuels was similar.

Strong scattering from soot aggregates affects both laser extinction measurements of soot concentrations and predictions of continuum radiation from soot. This is illustrated in Fig. 2, where present predictions \(\rho_{sa}\) are plotted as a function of wavelength. \(\rho_{sa}\) is significantly greater than unity for wavelengths

---

Research supported by the Building and Fire Research Laboratory of the National Institute of Standards and Technology, Grant No. 60NANB1D1175.
in the visible, which implies a corresponding overestimation of soot concentrations from laser extinction measurements interpreted using the Rayleigh scattering approximation.

Additionally, $\bar{P}_{sa}$ remains large for heavily sooting materials in the near infrared, where continuum radiation from soot is most important. The implication of these findings, as well as the properties of underfire soot, are currently being studied in this laboratory.

References


Fig. 1 Angular scattering pattern of overfire acetylene soot.

Fig. 2 Mean ratios of scattering-to-absorption crosssections for overfire soot.
ANCHORING MECHANISM OF POOL FIRES

S. Venkatesh, A. Ito and K. Saito
Combustion and Fire Research Laboratory
Dept. of Mechanical Engineering
University of Kentucky
Lexington, KY 40506

ABSTRACT: This paper describes that the entire flame sheet of a pool fire has a diffusion dominant structure and unlike jet diffusion flames a special premixing mechanism is not necessary to explain the anchoring mechanism near the base. In the case of momentum controlled turbulent jet diffusion flames the premixing near the burner rim occurs due to the presence of a shear-stress related circulation zone. We investigated the structure of the flame near the base of a 60 mm diameter pool fires of n-hexane and 1-propanol through detailed measurements of the velocity and temperature fields. Our measurements near the base of the pool fire show that the Reynolds shear stresses are two orders of magnitude less than that reported for a turbulent jet diffusion flame. A detailed investigation of the structure of the pool fire near the base indicates that the nature of the diffusion flame is similar to that in a counter-flow diffusion flame. The entrained air near the base and the rising fuel vapors create a stagnation plane and the fuel mixes into the oxidizer stream in the forward stagnation plane purely by diffusion and a flame sheet of finite thickness is established. It is shown that the upward tangential velocity of the fuel-oxidizer stream in the forward stagnation zone is lower than the stoichiometric flame-propagation velocity, thus allowing the pool fire to anchor.

Air entrainment into pool fires is a key phenomenon that determines the thermal fluid dynamic and chemical structure of the pool fire. A number of studies on the structure of pool fires have been conducted. Recent studies by Weckman et al. have provided detailed information on the structure of pool fires through simultaneous measurements of velocity and temperature profiles. Bouhafi et al. conducted measurements of temperature and species concentration on kerosene pool fires, and they suggest that air entrainment near the base leads to a convective-type mixing giving the flame a premixed character in this region. However, flame structure near the base is not clearly understood.

We studied in detail the mechanism of air entrainment, the thermal/fluid dynamic structure of the flame near the base and the flame anchoring mechanism. Detailed velocity and temperature profiles were measured near the base of pool fires of 1-propanol and hexane supported on a 60 mm diameter pan. We found that the pool fire can be divided into three distinct regions. Region 1—a persistent laminar blue zone formed near the flame base; Region 2—flame "neck" zone with wrinkled surface; and Region 3—intermittent zone above the "neck" zone. Our study was focused on the region near the base of the
pan where the flame is anchored.

The flame base structure determines the following two aspects: (1) Anchoring and stabilization of the pool fire. Common understanding for jet diffusion flames is that premixing occurs near the base and is responsible for anchoring and stabilization of the flame. Takahashi and Schmoll's results show mixing of the fuel and air through a circulation zone at the burner rim due to strong shear-stresses, leading to flame anchoring. Whereas, in a pool fire the fuel and oxidizer velocities near the base are much smaller, and insufficient to produce shear-stress-induced circulation zones. Therefore, the mechanism for fuel and oxidizer mixing in pool fires is different from jet diffusion flames. (2) Why and how much air entrainment occurs near the base? The air entrainment mechanisms near the base and in the intermittent regions are quite different. In the intermittent region of the flame, air entrainment is mainly by relatively large-scale buoyancy-induced mixing. Near the base, by contrast, air is convectively entrained to satisfy mass conservation because of the rapidly accelerating buoyant gases in the flame interior.

EXPERIMENTAL: The pool fires were established on a pyrex pan of 60 mm diameter. Velocity measurements were obtained using a two component phase doppler particle analyzer system, which consists of a 5 W argon-ion laser and standard transmitting, receiving and signal processing equipment. High speed data recording was achieved using a micro-computer, and data analysis is by datamanagement software of the system. Data recorded and analyzed includes instantaneous radial and axial velocities, mean and root mean square velocities and Reynolds shear-stresses. Outside the flame sheet, velocity measurements were made by seeding the ambient air with the talc particles. Inside the flame, flow profile was visualized using TiO2. Temperature measurements were made using a 75 μm-diameter chromel-alumel thermocouple.

SUMMARY: (1) Based on our experimental measurements on pool fires and finite rate chemistry concepts, the entire flame sheet of a pool fire is established to have a diffusion flame structure. The counter-flow diffusion structure of the flame at the base was established from a mass balance at the flame sheet, and through a comparison of the location of the flame sheet in the convective-air entrainment zone for fuels with different stoichiometric fuel-air requirements. (2) Air entrainment through the quenching zone was found to be a small fraction of the net air entrainment near the base. (3) The fluid dynamic structure of the anchoring mechanism of a pool fire and a jet diffusion flame is different. In a jet diffusion flame, Reynolds's shear-stress near the rim of the burner induce a stagnant circulation zone where the fuel and the oxidizer are mixed and the flame anchors. Contrary, in a pool fire the shear-stresses at the rim are two orders of magnitude lower compared to the jet diffusion flame, and turbulent mixing does not occur. Finite-rate chemistry establishes the presence of a molecular-diffusion mixing zone that allows the flame to anchor at the base.
EFFECT OF PRESSURE AND OXYGEN CONCENTRATION
ON PAN FIRES IN AN ENCLOSED SPACE

Jean L. Bailey, Patricia A. Tatem, and Frederick W. Williams
Navy Technology Center for Safety and Survivability
Naval Research Laboratory
Washington, D.C.

Large-scale fire tests were conducted in the Naval Research Laboratory’s 324 m³ pressurizable fire test facility, FIRE I (Alexander et al., 1982) to determine the effect of pressure and oxygen concentration on 0.5 m diameter methanol pan fires in a totally enclosed space. Data from 14 experiments were analyzed in support of this study. Eight were conducted with an initial chamber atmosphere of 21 vol-% oxygen and one atmosphere pressure. The location of a moveable thermocouple array, designed to make detailed temperature measurements of the chamber interior, was varied during these eight experiments. The remaining six experiments began with the chamber interior as described in the table below.

<table>
<thead>
<tr>
<th>Test</th>
<th>Initial Oxygen Conc. (Vol %)</th>
<th>Initial Pressure (PSI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>M425B</td>
<td>17</td>
<td>14.7</td>
</tr>
<tr>
<td>M430A</td>
<td>15</td>
<td>14.7</td>
</tr>
<tr>
<td>M430B</td>
<td>15</td>
<td>14.7</td>
</tr>
<tr>
<td>M502A</td>
<td>21</td>
<td>23.9</td>
</tr>
<tr>
<td>M503A</td>
<td>21</td>
<td>17.6</td>
</tr>
<tr>
<td>M514A</td>
<td>21</td>
<td>20.7</td>
</tr>
</tbody>
</table>

After the desired conditions were reached within the chamber, the chamber was sealed and the pan fire was ignited. The pan fires in all of the experiments with the exception of M425B, M430A, and M430B were allowed to burn for one hour before they were manually extinguished. The M425B, M430A, and M430B panfires continued to burn until they self-extinguished. Mass loss rate, chamber gas composition (oxygen, carbon dioxide, and carbon monoxide), pressure, and chamber gas and wall temperatures were among the measurements recorded.

Figure 1 shows mass loss rate as a function of time for the reduced oxygen tests. M425A is shown representing the eight experiments which had initial oxygen concentrations of 21 vol-% and initial pressures of one atmosphere. The lower the initial oxygen concentration, the smaller the initial burning rate. This order was maintained.
throughout the experiments as the mass loss rates decreased because of diminishing oxygen within the chamber.

Mass loss rates versus time for the enhanced pressure experiments are shown in Figure 2. Again, M425a is representing the experiments conducted at 21 vol-% initial oxygen concentration and one atmosphere pressure. Higher initial burning rates correspond to higher initial chamber pressures. The mass loss rates decline with time as a result of the decreasing oxygen concentration.

Analysis of the data revealed a linear relationship between the mass loss rate and oxygen concentration between 21 vol-% and 14 vol-% oxygen concentration. Below 14 vol-%, the mass loss rate rapidly declined until the fire self-extinguished at approximately 12 vol-%. Further data analysis relating mass loss rate to chamber pressure and oxygen concentration will be discussed.

REFERENCES:

Fire-Induced Winds in the 1991 Oakland Hills Conflagration

Javier Treles
Patrick J. Pagni

Mechanical Engineering Department, University of California, Berkeley, CA 94720

The tragedy of the October 20, 1991 Oakland Hills “Tunnel” fire has brought into sharp focus the role of wind in conflagrations. The effects of hot, dry, fast atmospheric winds (called “Diablo Winds” in this area) are well known. The magnitude of the fire-induced winds is not well understood and is the subject of this paper.

1.0 Theory

This research is based on the model developed by Howard Baum and Bernard McCallroy [1]. The total convected heat release rate $Q_o$, the ambient density $\rho_o$, the ambient temperature $T_o$, the specific heat $c_p$, and the acceleration of gravity $g$ are used to determine characteristic quantities with which to nondimensionalize all physical quantities. The expressions for the characteristic length $L$, velocity $U$, vorticity $\omega$, potential $\Phi$, and Stokes stream function $\Psi$ are given in eq. (1).

$$L_c = \left( \frac{Q_o}{\rho_o c_p T_o g} \right)^{2/5} \quad U_c = \left( \frac{3^2 Q_o}{\rho_o c_p T_o} \right)^{1/5} \quad \omega_c = \left( \frac{3^2 \rho_o c_p T_o}{Q_o} \right)^{1/5} \quad \Phi_c = \frac{3^{10} Q_o}{\rho_o c_p T_o} \quad \Psi_c = \frac{Q_o}{\rho_o c_p T_o}$$

All the subsequent nondimensional quantities - superscripted with an asterisk - are obtained by dividing the physical quantity by the characteristic one. Once one has solved for the flow field induced by a single fire, that produced by a series of fires is determined by physically scaling each single fire field and then vectorially adding all the influences at a point.

The nondimensional flow field in axisymmetric cylindrical coordinates is decomposed into an expansion velocity $u^*$ governed by a potential $\Phi^*(r^*, z^*)$ and a solenoidal velocity $v^*$ described by a stream function $\Psi^*(r^*, z^*)$. The expansion field is determined by solving a Poisson equation (2) for $\Phi^*$ with a normalized Gaussian heat release rate from a single fire $Q^*(r^*, z^*)$ as a forcing function. The boundary conditions subscripted by $Q$ refer to the asymptotic value. For the potential, this is simply the Green’s function for a point source.

$$\frac{1}{r^*} \frac{\partial}{\partial r^*} \left( r^* \frac{\partial \Phi^*}{\partial r^*} \right) + \frac{\partial^2 \Phi^*}{\partial z^2} = \left( \frac{4.25}{\pi} e^{-2.18r^*} \right) \quad \text{if} \quad z^* \leq 1.32$$
$$= 0 \quad \text{if} \quad z^* > 1.32$$

$\Phi^*(0, z^*)$ is finite $\quad \Phi^*(10, z^*) = \Phi^*_a(10, z^*)$ $\quad \frac{\partial \Phi^*}{\partial r^*}(0^*, 0) = 0 \quad \Phi^*(r^*, 20) = \Phi^*_a(r^*, 20)$

$$\Phi^*_a(r^*, z^*) = \left[ \frac{4\pi^2 (r^2 + z^2)^2}{\pi} \right]^{1/2}$$

The stream function is determined by solving a different Poisson equation (3) which has the vorticity $\omega^*_e(r^*, z^*)$ as the inhomogeneity. Empirically determined profiles for the center-line temperature $\Theta^*(z^*)$ and vertical velocity $U^*(z^*)$ [2] are employed to determine $\omega^*_e$. $\omega^*_e$ was derived by taking the curl of the velocity obtained from a Gaussian model which takes compressibility effects into account via the energy integral $I_\lambda$ (4). The quantity $R^*$ is the “width” of the Gaussian profile. $I_\lambda$ is determined using the Romberg numerical quadrature routine gromb detailed in [3] for $\lambda = 0.866$. Accuracy is verified by calculating $I_\lambda$ for $\lambda = 1$ and comparing with the exact logarithmic answer.

$$r^* \frac{\partial}{\partial r^*} \left( \frac{1}{r^*} \frac{\partial \Psi^*}{\partial r^*} \right) + \frac{\partial^2 \Psi^*}{\partial z^2} = -r^* \omega^*_e(r^*, z^*) = -r^* \left[ \frac{2r^* U^*(z^*)}{R^2(z^*)} e^{-\left( \frac{r^*}{R(z^*)} \right)^2} \right]$$

$\Psi^*(0, z^*) = \Psi^*_a(r^*, 0) = 0 \quad \Psi^*(10, z^*) = \Psi^*_a(10, z^*)$ $\quad \Psi^*(r^*, 20) = \Psi^*_a(r^*, 20)$
\[ R(\xi^*) = \left[ \pi U(\xi^*) \left[ 1 - I_\lambda(\psi(\xi^*)) \right] \right]^{-1/2} \]

\[ I_\lambda(\psi(\xi^*)) = \int_0^1 \frac{dt}{\left( 1 + [\psi(\xi^*)] [t^{1/2}] \right)} \]

The form of the asymptotic stream function \( \Psi^* \) is complicated. It is convenient to first convert to a polar coordinate system \((\xi, \theta)\) \((5)\). This transformation is applied to equation \((3)\). In order to determine the asymptotic stream function, the expression for the vorticity is replaced by its asymptotic value \((6)\). The asymptotic stream function \( \Psi^* \) is of the form \((7)\).

\[ \xi = \sqrt{r^* + z^*} \quad \theta = \tan^{-1} \left( \frac{r^*}{z^*} \right) \quad \mu = \cos \theta \quad x = \frac{1 + \mu}{2} \]

\[ \omega^* (r^*, z^*) \sim \xi^{4/3} \Omega (\theta) \text{ as } (\xi \to \infty) \quad \text{where} \quad \Omega (\mu) = -\frac{6\pi^2 B^2}{7 \mu^3} e^{-\frac{3\pi AB}{7} \left( \mu - \frac{1}{\mu^2} \right)} \]

\[ \Psi^*_a (r^*, z^*) = \xi^{5/4} G(\theta) \quad \text{where} \quad \frac{d^2 G}{dx^2} + \frac{10}{9x(1-x)} G = 4\Omega(x) \quad \text{with} \quad G(0.5) = 0 \quad G(1) = 0 \]

The differential equation in \((7)\) was solved via the collocation boundary value problem solver \textit{colnew} from ODE. \( \Psi^* \) is determined numerically using the fourth order \textit{FISHPACK} routine \textit{sepx4}. \( \theta^* \) is calculated using the centered finite difference solver \textit{nhscyl} also from \textit{FISHPACK}. For both \( \Phi^*_a \) and \( \Psi^*_a \), \( \Delta z^* = 0.1 \) were used. In order to determine the radial velocity \( v^*_r \) and the axial velocity \( u^*_z \) \((8)\), appropriate numerical derivatives of the potential and stream function are taken and summed yielding the complete flow field for a single fire.

\[ u^*_r = \frac{\partial \Phi^*}{\partial r} - \frac{1}{r} \frac{\partial \psi^*}{\partial z} \quad u^*_z = \frac{\partial \Phi^*}{\partial z} + \frac{1}{r} \frac{\partial \psi^*}{\partial r} \]

Once the nondimensional flow field for one fire has been obtained, the velocities induced by \( N \) fires are found by scaling the calculated flow to physical dimensions which depend on ambient conditions and the heat release rate of the fire \((x_1, y_1)\). To obtain the component velocities induced at a grid point \((x_j, y_j)\), simply vectorially sum all the velocities induced by the surrounding \( N \) fires at this point. For very large fires, a sub-grid model is used. One directly sums all the velocities induced at a point by fires within the local cell. Fires in each cell outside this cell are approximated by a single fire centered at the heat release rate centroid of their cell. The summation is then made over the velocities induced by the centroid fires of the outside cells.

20 Results

The radial and vertical velocities induced by the burning houses and vegetation will be determined at three times during the fire (11 AM, 1 PM, and 4 PM) at several altitudes. The heat release rate of the burning structures will be modelled as 30 MW for the first hour of burn and 1 MW for the next six. The resulting velocities will be vectorially added to the mean ambient wind. The complete flow field helps to explain how a configuration can quickly propagate. The fire induced winds will only subside when the fuel load is consumed. The wind that rushes in at the bottom emerges from the plume hot and laden with burning brands. In the fire of October 20, 1991, this burning debris was conveyed by the plume into a strong Diablo Wind which sent these ignition sources to fresh areas with extremely dry fuel, propelling the fire rapidly downwind.

30 References


PAPER WITHDRAWN
PAPER WITHDRAWN
A LUMPED PARAMETER MODEL for UNBURNED TREE CROWN STREETS

M.C. Smith
Michigan State University
Warren G. Heilman
U.S. Forest Service, East Lansing, Michigan

Strips of unburned tree crowns occur occasionally in wildland forest fires. These are called unburned tree crown streets in the literature and remain an unexplained phenomenon. Interest in these streets is based on the fire induced airflow and ambient winds which may produce the effect and ultimately on the hope that understanding will lead to better and safer fire fighting procedures.

References (1) through (6) contain observations and discussion of the phenomenon, some of which include possible explanations of the causes of the appearance of the streets. Present explanations include horizontal roll vortices, wind shifts, firebrand throwing and buoyancy induced wind flow. None of these is entirely satisfactory due to the complexity of the three-dimensional transient problem involving fluid mechanics, heat transfer and combustion.

The buoyancy induced wind flow model presented here is believed by the authors to account for most of the observed effects and offers a plausible explanation of the phenomenon. Confirmation requires further observation and perhaps instrumentation and data gathering.

The induced air flow model yields Equation (1)

\[ m \frac{dv}{dt} = \text{buoyant force} - \text{resistant force} \]  

where \( m \) is the gas mass in the air flow column, \( v \) is the average induced flow speed, \( t \) is time and the buoyant and resistant forces are

\[ \text{buoyant force} = g \beta \Delta T \rho A_c H \]  
\[ \text{resistant force} = c \rho v^2 \]

where \( g \) is gravity, \( \beta \) is the coefficient of thermal expansion, \( \Delta T \) is the temperature rise due to combustion, \( \rho \) is air density, \( A_c \) is flow-crossection area, \( H \) is height of the flow column, \( c \) is a drag coefficient and \( v \) is the flow speed.

The equation provides an oscillatory solution on the basis of heat release from burning tree crowns being substantially larger than heat release from understory burning.
The effect is to increase and decrease the induced flow periodically. This oscillation produces the unburned tree crown streets.

It is evident that conditions must be favorable within a narrow range for the phenomenon to occur, which may account for the relative rarity of unburned tree crown streets occurring in wildland forest fires. Results of calculations show that the conditions for oscillation to occur depend upon the ratio of understory to crown heat release rates, a dimensionless induced inflow air speed, and the ratio of the crown base height to the maximum understory flame length.

Analysis was by digital computer using numerical integration and data from References (6) through (9). Additional results obtained indicated that the major effects on the oscillation are accounted for by the use of these three dimensionless parameters.

Other explanations of the occurrence of unburned tree crown streets cannot be ruled out. A more sophisticated and detailed model and the establishment of a wider range of data are desired to consider predicting the occurrence of the phenomenon. Results may then be reviewed for application to forest fighting and prediction.

REFERENCES

Numerical Simulation of the Rise and Dispersion of Fire Generated Plumes

Ahmed F. Ghoniem and Xiaoming Zhang
Department of Mechanical Engineering
Massachusetts Institute of Technology
Cambridge, MA 02139

Abstract

The rise and dispersion of plumes generated by massive fires are important phenomena which determine the environmental impact, hazards potential and possibly extend of fire spread. This is because massive fire plumes are strongly buoyant, are laden with unburnt hydrocarbon and carbonous particulates, and can induce entrainment currents of very high velocities. Because of their strong buoyancy, these plumes can penetrate the Planetary Boundary Layer velocities and interact dynamically with density its stratification. The latter complicates the modeling of large plumes but leads to a rich variety of phenomena which, in order to accurately assess the environmental impact of massive fires, must be examined in detail.

We have developed a moving surface formulation for the problem of buoyant plume dispersion in a stratified atmosphere, based on the parabolization of the governing equation in the major wind direction (Ghoniem et al., 1992). The solution, including the plume trajectory, the evolution of the plume cross sections, and the plume-induced flow is obtained using the transport element method. This is a Lagrangian vortex method extended to include the dynamic effects of density stratification. Our computational plume model thus is an efficient large eddy simulation, in which small scale atmospheric turbulence is modeled using a new interpretation of the classical K-theory. The model is flexible enough to accommodate other physically relevant processes such as chemical reactions between the plume material and the atmosphere. Solutions are compared with experimental measurement whenever possible.

Figure 1 shows the density distribution within the plume cross section for three different buoyancy Reynolds numbers defined such that the velocity is that induced by gravity and the diffusivity is the combined molecular and turbulent diffusivities. In all cases, atmospheric density is uniform. Clearly the large scale structure of the plume is not strongly affected by small scale diffusivity. This structure, while strongly deforming by the action of two counter rotating streamwise vortices which form as the gravity generated vorticity on the plume boundary rolls-up, indicates that the dispersion of the plume material during the rise phase is rather limited. In all cases, the experimentally observed kidney shape of the plume cross section is predicted by the numerical simulation (Fanaki, 1975). The effect of the Reynolds number on the plume trajectory, and a comparison with the widely accepted two-third power law (Briggs, 1975, and Weil, 1988) are shown in Fig. 2. As found experimentally, the model shows that eddy viscosity, or small scale turbulence, has no effect on the plume trajectory (Hewett et al., 1971).

The effect of atmospheric density stratification, or inversion is shown in Fig. 3. During the daytime, plume rise takes place in a convective boundary layer capped by a sharp inversion which stops the plume from further penetration, except of its buoyancy is exceedingly strong or if the inversion layer is very close to the ground (Manins, 1979). This is shown by the results in Fig. 3 where the plume penetrates a low weak inversion layer in (a); partially penetrates an elevated weak inversion layer in (b), and is prevented from penetration by a low, strong inversion layer in (c). In case (b), only the middle section of the plume, which maintains the strongest buoyancy can penetrate the inversion while the side parts are reflected back. In the last case, all the plume material are reflected off the inversion and sent back downwards.

References


---

**Figure 1.** The density contours in the plume cross wind section at $x=5$ for different buoyancy Reynolds numbers. Only half of the cross-section is shown due to the symmetry about the center line.

**Figure 2.** The rising plume trajectories for various buoyancy Reynolds number. The solid line is the extended "two-thirds" law which includes the effect of the finite initial plume size. Plume trajectories corresponding to cases (a), (b) and (c) in Figure 3 are also included.

**Figure 3.** The density contours at successive downwind location for a plume approaching an inversion layer when: (a) the plume is initially close to a weak inversion layer (penetration); (b) the plume is initially far from a weak inversion layer (partial penetration); and (c) the plume is initially close to a strong inversion layer (reflection).
Two-Dimensional Simulations of Buoyancy-Induced
Horizontal Roll Vortices

W. E. Heilman
USDA Forest Service, East Lansing, MI 48823

The intense heat associated with wildland fires can generate very strong atmospheric boundary-layer
circulations in the vicinity of burning regions. Hot gases produced within a wildland fire lead to buoyancy-induced
upward vertical velocities over the fire and low-level horizontal flow into the burning area. The buoyancy effect
over and adjacent to firelines in wildland fire episodes is primary mechanism involved in the formation of
atmospheric horizontal roll vortices, as outlined by Church et al. (1980). Evidence of horizontal roll vortices
during wildland fire episodes has been documented by Haines (1982), and these vortices have been generated in
numerous wind-tunnel experiments (Haines and Smith, 1987, 1992). The presence of horizontal roll vortices can
influence the movement of flames, firebrands, and smoke, all potential hazards for fire-fighting personnel. In
addition, horizontal roll vortices have been indicated as a possible factor in the formation of unburned tree-crown
streets within burned forested areas (Haines, 1982).

In this study, the buoyancy mechanism for horizontal roll vortex development is examined through the use of a
two-dimensional, nonhydrostatic, atmospheric boundary-layer model. Prognostic equations are used to solve for
the horizontal and vertical velocity components, the potential temperature, and the turbulent kinetic energy.
Nonhydrostatic dynamic pressure perturbations are diagnosed from a Poisson equation derived from the
atmospheric continuity equation. The governing equations are numerically solved by both finite-element and
finitedifference techniques.

The presence of single and multiple firelines is approximated by introducing constant surface potential
temperatures of 900°K or 1500°K at prescribed locations in the modeled domain, which spans a 20 km wide by 1.8
km high vertical plane. Initial potential temperatures at all other locations in the modeled domain are set at
298°K. In order to simulated the impact of varying crossflow wind speeds on the development of horizontal roll
vortices, different initial logarithmic crossflow velocities are introduced into the model simulations. Simulations
are carried out for a period of 90 s, with a time step of 0.05 s and 1800 time steps per simulation.

Model simulation results indicate that the development and strength of horizontal roll vortices are influenced
by the temperature of the approximated firelines, the proximity of the firelines to each other, the presence of light
ambient winds blowing perpendicular to the firelines, and the slope of the underlying terrain. For a single fireline
located on flat terrain under calm crossflow conditions, a symmetric vortex pair develops over the fireline. Larger
surface temperatures at the fireline produce more vigorous vortex circulations. The presence of a light ambient
crossflow completely inhibits the development of these symmetric vortices for the temperature scenarios performed
in this study. However, the presence of a sufficiently light ambient crossflow results in the development of low-
level vortices with significant vorticity immediately downwind of single firelines. If this type of buoyancy-induced
vortex were to develop along the flank of an actual fireline, with flames and firebrands caught in the circulation,
fire-fighters could be endangered. An analysis of the nonhydrostatic dynamic pressure perturbations near the
region of surface heating reveals that higher surface temperatures generate larger pressure perturbations because of
the increased low-level convergence over the firelines. Turbulence energy over individual firelines tends to
diminish as the ambient crossflow speed increases and the fireline temperatures decrease.

When multiple firelines are present, interesting interactions between adjacent horizontal roll vortices are
observed. During calm crossflow periods, buoyancy-induced horizontal roll vortex pairs develop over each fireline
if they are sufficiently far apart from each other. Numerical simulations indicate a minimum fireline spacing for
vortex development to be on the order of the characteristic horizontal dimension of the vortices. The presence of
multiple firelines increases the likelihood of vortex interaction and downdraft reinforcement between the firelines.
Downdraft wind speeds between firelines were observed to increase by nearly 40% over the speeds simulated with
only one fireline present. Downdraft reinforcement by adjacent horizontal roll vortices in actual wildland fire
episodes could inhibit the burning of tree crowns and aid in the development of unburned tree-crown streets.
Horizontal roll vortex development in the presence of light ambient crossflows over multiple firelines spaced
sufficiently far apart is much more prominent than for individual firelines. Upwind firelines tend to act as "barriers" to the flow for the downwind firelines (see Fig. 1).

Numerical simulations also indicate the presence of significant low-level horizontal flow directly over the outermost firelines and directed toward the innermost firelines. This is due to the thermal characteristics of the boundary layer inside and outside of the region bounded by the firelines. This phenomenon is more pronounced when the firelines are closer together. In an actual wildland fire with multiple firelines and an ambient flow parallel to the firelines, this phenomenon suggests a migration of flames and firebrands toward inner firelines and the creation of curved unburned tree-crown streets in response to the downdraft segments of the accompanying migrating roll vortices.

The effect of terrain irregularities on boundary-layer circulations has been well documented. Sloping terrain also affects buoyancy-induced circulations associated with firelines. Present numerical simulations indicate that symmetric horizontal roll vortices are less likely to develop on the slopes of simple hills under calm crossflow conditions than over flat terrain. However, firelines located on the crests of simple hills generate horizontal roll vortices that become stronger as the steepness of the hills increases. Vortex activity associated with firelines on simple hills is very dependent on fireline location and terrain steepness when ambient crossflow wind speeds increase. The intensity of horizontal roll vortices associated with firelines on the crests and leeward slopes of simple hills is greatly diminished when ambient crossflow winds increase in speed, especially over steeper hills.

![Figure 1](image_url)

Figure 1. Simulated twodimensional wind fields resulting from a surface potential temperature of 1500°K at \( x = 800 \) m, \( x = 1000 \) m, and \( x = 1200 \) m with (a) no ambient crossflow and (b) a light ambient crossflow corresponding to an initial friction velocity of 0.1 m s\(^{-1}\).

References


Experimental Study of Forced Ventilation Glovebox Fires

Michelle J. Peatross and Richard J. Roby
Mechanical Engineering Department
Virginia Tech
Blacksburg, VA 24061

Craig L. Beyler
Hughes Associates, Inc.
6770 Oak Hall Lane, Suite 125
Columbia, MD 21045

Introduction

Gloveboxes are widely used to allow processing hazardous materials without the need for direct exposure of workers to the materials. Fires within such gloveboxes can result in the loss of containment of the hazardous material and result in exposure of workers to dangerous materials. As such there is a need to limit combustible materials in gloveboxes and assure that the glovebox is designed to withstand the fires which may occur without serious loss of containment. The weak points in most gloveboxes are the gloves and the window assemblies. The most serious fire threats in gloveboxes are combustible liquids.

A test series was carried out in which a glovebox mockup was exposed to a range of combustible liquid pool fires within the box. Due to the relatively low ventilation rates, these fires were primarily underventilated fires. These underventilated fires are unique in that the fire occurs in a forced ventilation compartment fire environment with ventilation provided from the top of the glovebox. This ventilation geometry is not expected to give rise to a well defined two layer fire environment [1,2]. The tests were designed to provide insights into the failure of glovebox components and to develop an understanding of forced ventilation, underventilated compartment fires.

Experimental Apparatus and Procedures

A glovebox mockup with overall dimensions of 12’ X 4’ x 4’ was built and instrumented. The box consisted of an angle iron frame with steel panels. Four ceiling and four wall window assemblies were installed in the glovebox along with four pairs of gloves along the 12’ length of the mockup. All window gaskets were floating type gaskets constructed of neoprene. The gaskets were partially protected by steel covers designed to prevent the window from falling into the glovebox if and when the gasket burned out. This protective gasket cover is known as a fireguard. Ceiling windows were 1/4’’ wired glass and wall windows were 1/2’’ laminated lead glass windows. Gloves were constructed of neoprene.

Four fuel pans were used in the test series: 3 X 3’, 33’’ diam., 24.5’’ diam., and 11’’ diameter. All pans were positioned in front of a window/glove pair location. The fuel used was a hydrocarbon combustible liquid. Air flows to the glovebox was via a 6’ duct in the roof of one end of the glovebox, and exhaust was via a 6’ duct on the roof at the opposite end of the 12’ long box. The HVAC system was operated to provide a negative 3/4’ of water inside the box while maintaining a flow of 200 cfm through the glovebox.

The glovebox was instrumented to measure the fuel pan mass, gas and window temperatures, heat fluxes, and exhaust gas flow and composition. Gas temperature measurements were made with an aspirated thermocouple tree with a TC spacing of 6’. Floor, wall, and ceiling heat fluxes were measured. Exhaust gases were analyzed for carbon dioxide, carbon monoxide, oxygen, and unburnt hydrocarbons.

Results

Open pan burning rates and the glovebox ventilation rates indicated that all fires would be underventilated. The heat release rate indicated by the ventilation rate was about 150kW and the open
pan burning rates were as high as 1.5-2 MW. Flames observed in the 3' X 3' pan were highly unsteady and did not cover the full surface area of the pan. The flame was sufficiently unstable that it often extinguished prior to consuming all the fuel. Other pans burned until the fuel was consumed.

The thermal environment created in the tests was most severe for a two hour 3' X 3' pan test. Temperatures up to 600°C were observed in this test near the ceiling. The temperature profile over height was relatively linear as had previously been observed in compartment fires ventilated from above [1-4]. The gas temperatures near the floor were 300°C rising linearly to 600°C near the ceiling. Such conditions were maintained for as long as two hours in one test. Observations of the performance of the glovebox system will focus on this worst case exposure. In the first few minutes of the experiment both the wired and laminated glass windows cracked. The first to crack was the ceiling window above the fire and the first wall window to crack was also adjacent to the fire. However, neither the wired nor laminated glass ever broke out. Only one window gasket burned out during the test and the window was maintained in place by the fireguard. Gloves were damaged by the fire especially those gloves directly exposed to the pan fire. When the hand of the glove was burned off, the remainder of the glove stranded from the heat. This resulted in a reduction in the size of the glove hole. Gloves not inserted into the box, i.e. in the storage position were not damaged sufficiently by the fire to create any openings.

Containment of the fire gases in the compartment was maintained despite the glove leaks, except during puffing of the fire. During a puff, smoke was intermittently lost through the glove holes. The total volume of gases emitted was a small fraction of the total flow of fire gases from the glovebox. While fire extinguishment was not pursued in the test series, the tenuous nature of the flames indicated that extinguishment with portable extinguishers via a glove hole would be quite straightforward for trained personnel.

While these pans would support far greater heat release rates than the ventilation rate would allow, the actual fuel volatilization rates and resulting heat release rates were on the order of 100 kW. The extent of underventilation of the fire was strongly reduced by the reduction in fuel volatilization caused by limited ventilation. The evidence was that the larger pans were modestly underventilated while the smaller pans were overventilated. These observations are based on CO and total hydrocarbon measurements in the exhaust.

Despite the fact that the downward forced ventilation caused a distinctly non-two layer fire environment, the CO levels observed in the exhaust were similar to CO levels previously observed in very well defined two layer environments [5].

References:


Corridor Flow of Smoke from Postflashover Fire: a post-"FIRE SIMULATOR" Module for FPETOOL

by
Harold Nelson and Scot Deal
NIST

In a significant number of multi-death fires in health care occupancies, the deceased were found in locations remote from the room of fire origin. Hot smoke from the fire travelled between 3 and 30 meters (10-100 feet) before reaching the victims. To better understand the mechanisms which caused these far field deaths it is necessary to more accurately predict smoke properties when the smoke is near the victim. Therefore, it is also necessary to predict the smoke properties as it travels from the room of fire origin and spreads throughout the building. This model focuses on predicting the properties of smoke as it travels away from a flashover room fire and proceeds down a corridor.

Funding for this project was provided by our generous sponsors, the General Services Administration and the administration of the Building and Fire Research Laboratories, NIST (whose tithing of congressionally earmarked money we could not do without). The effort was collaboratively produced by the two authors, whose names appear with respect to their contributions.

The "Corridor" simulation (as this smoke flow simulation will now be called) is a first-order engineering method at predicting conditions in the smoke from a postflashover fire as it progresses down a corridor away from the room of fire origin. Corridor simulates smoke displacement with a first order, nonlinear, homogeneous ordinary differential equation. The solution was approached using first order Runge-Kutta or Euler's method in pseudo steady state. The frame of reference was Eulerian, but because of heat transfer (as will be explained) the smoke temperature cools with age, hence temperature is mildly dependent upon time in addition to its explicit dependence upon position. Generous assumptions and hence limitations were imposed upon the model for purposes of simplification and to make appropriate delivery, however, if this project had been easy--it would have already been completed.

Corridor's first assumption is that the fire is postflashover. The limitation this imposes effects not temperature, but smoke flow. The program operates with a constant rate of smoke flow from the room of fire origin. The smoke flow rate is most readily obtained by the user from a room fire model (such as FIRE SIMULATOR or FAST); however, smoke flow rates are one of the least accurately predicted quantities from room fire models. The temperature of the smoke can be accounted for down to to levels below 250°C, but this temperature must be constant with time for the duration of the simulation.

The fire gases issuing into the corridor entrain air if they have any substantial horizontal or vertical momentum. Corridor simulates vertical entrainment, not horizontal. As the height through which the issuing gases entrain ambient corridor gases increases, the smoke filling becomes qualitatively more and more similar to a zone room model. The user of Corridor is notified at a point where
this two zone filling behavior should begin to occur. The point is identified in corridor when the entrained air to smoke mass ratio exceeds 3.39. Gas concentrations and enthalpies are appropriately adjusted for ambient air entrainment.

Heat is lost from the smoke to the ceiling and walls of the comdor. This is accounted for in an implicit, exponential relationship identified by Heseldon from experimental observations. Comdor accounts for the corridor surface area which contacts the advancing smoke wave. The impact of this heat loss on smoke temperatures is an exponential decay with distance and a modest cooling with time.

The depth of the corridor smoke is predicted from the volume and horizontally projected area of the wave. The volume is obtained by dividing the cumulative smoke mass flow into the corridor by the average smoke temperature. The average smoke temperature is determined from an average integrated over the distance of the wave.

Results from Comdor are included in a soon to be published NISTIR by Nelson and Deal. This work brought to light several items of interest. First, if experiments were performed wherein the temperature of the flashover smoke from the burning room were increased with each successive experiment, we would observe, quite intuitively, an increase in the velocity of the advancing smoke. The depth of the smoke would decrease though. Apparently, the effect of increased travel speed overcomes the effect of density and increased voluminity. The second item arose from applying Comdor rather than from intrinsically using Comdor. The refuge room proximity to the room of flashover makes little difference on occupant survival. This curious fact arises from the effect of density. Refuge rooms close to the fire are exposed to hot comdor gases, however, the mass of hot comdor smoke leaking into next-door refuge rooms is not substantially greater than the mass of cool smoke leaking into refuge rooms located at the end of the hall. Smoke at the end of the hallway is much cooler, but also much denser.

Additional features about Comdor

1. There is no consideration of depositional mass loss from the smoke wave.
2. Smoke flow can be split in two, to simulate a room located in the middle of a corridor.
3. The heat loss factor can be user modified to simulate corridors of different frictional characteristics and to simulate different thermal properties of corridor lining materials.
4. Corridor asks for smoke level in the opening, but most fire models give only the smoke interface level in the room.
5. There is no consensus on door jet entrainment rates.

2. Personal Communication with Baum, H., NIST, Gaithersburg, MD, USA May 1992.
Flame Retardancy of Cotton - Synthetic Blend Fabrics

Robert J. Harper Jr., George F. Ruppenicker and A. P. S. Sawhney
Southern Regional Research Center, USDA
New Orleans, La.

ABSTRACT

It has long been recognized that the treatment of cotton-polyester or cotton-nylon blends for flame retardancy is more difficult than the treatment of 100% cotton fabrics. The reason is that while the 100% synthetic can melt, drip, or shrink away from a flame, in a blend the composite forms a grid that prevents the synthetic from dripping away from the flame. This presentation reviews approaches for making cotton-blend fabrics flame retardant (FR), describes an unexploited approach for making these fabrics FR (1,2) and reports on the application of flame retardant finishes to a variety of blend fabrics. The usual approaches for treating blend fabrics utilize an antimony-halogen system, in which a halogen component such as decabromodiphenyl oxide is retained on the fabric with a polymer binder or in which a halogen containing polymer is employed as the halogen component (3,4). The unexploited approach is likewise based upon an antimony-halogen system. In this case, the halogen component is added to the fabric as of a reactive additive, which is fixed on the fabric via a resin system. The reactive halogen employed is dibromoneopentyl glycol. This approach leads to fabrics with durable press performance as well as FR performance. If the synthetic is a minor component of the blend, phosphorous-nitrogen (P/N) systems can be used to make the blends FR. Blends in which the synthetic component is inherently flame resistant such as a cotton-fiberglass blend can be readily made FR with finishes normally used on all-cotton fabrics.

The blend fabrics studied will include intimate blends as well as fabrics in which the synthetic is present as a filament core with a cotton wrap on the outside (5,6). Fabrics made from a combination of cotton and filament core yarns offer advantages not only in comfort, but also in ease of finishing. A factor with filament core fabrics is that the extra strength conferred by the filament component permits the use of a smaller percentage of synthetic in the fabric, thus reducing the amount of finish add-on required for FR. The influence of the change in fiber content, location of synthetic in yarn, and finish strength on FR performance will be noted.

Finishing of these special blend fabrics was primarily done using the precondensate - NH3 finish (7,8). The precondensate is prepared from tetrakis (hydroxymethyl) phosphonium chloride and urea. The fabric is padded with the precondensate, dried, and given a cure with ammonia gas. FR performance is measured by the vertical flame test as well as by Oxygen Index. In addition to the usual cotton-polyester blends, the finishing of other blends of cotton with fiberglass, FR nylon, and aramids will be included (9-11). The finishing a fabric containing an intrinsic FR synthetic component verses a non-FR synthetic component in the blend will be discussed. Fabrics used in this study were designed for protective clothing as well as for usage in outdoor fabrics (tents).
REFERENCES

DIRECT MEASUREMENT OF HEAT RELEASE RATES.

C. VOVELLE, J.L. DELFAU, F. MOUSSAN
Laboratoire de Combustion et Systèmes Réactifs - CNRS
1C Avenue de la Recherche Scientifique, 45071 ORLEANS CEDEX 2 France.
C. PHAM VAN CANG
EDF Direction des Etudes et Recherches, 1 Avenue du Général de Gaulle, 92141 CLAMART CEDEX.

INTRODUCTION

It is now widely accepted that heat release rate is a crucial parameter to characterize the flammable hazards of material (1, 2), and various calorimeters have been developed to measure this quantity. Laboratory or commercial apparatuses fall into two groups: (a) direct measurement (b) calculation from the amount of oxygen consumed. It is frequently reported that the first group methods suffer from heat losses and thermal inertia so that the oxygen consumption method is considered as more promising. However the use of a constant value for the heat released per unit mass of oxygen consumed is questionable for composite materials, especially for materials with fire suppressant included.

In this abstract we describe a new calorimeter that allows heat release measurements with both techniques. Emphasis has been put on the measurement of both radiative and convective heat components, so that heat losses are negligible and a direct comparison between both measurement techniques can be performed.

A preliminary study has been carried out with a methane diffusion flame to check the thermal inertia of the apparatus and to obtain heat release rate data in well defined conditions. To demonstrate the ability of this device to measure heat release rates of solid materials, experiments have been performed with PMMA and PVC samples as well.

EXPERIMENTAL

The calorimeter is derived from the Tewarson's flammability apparatus (3) with, as a major change, a water cooled stainless steel chamber surrounding the sample. A first component of the heat release rate ($Q_{T1}$) is derived from the water flowrate and the temperature difference between the ends of a water coil tube solded around the the stainless steel chamber. Combustion gases are mixed with air and collected in an exhaust chimney and a second heat component ($Q_{T2}$) is calculated from air flowrate and temperature.

Gaseous samples are withdrawn in the exhaust duct and introduced into continuous analysers that measure $O_2$, $CO$, $CO_2$ and total hydrocarbon concentrations. A second value of the heat liberated during combustion ($Q_{O2}$) is obtained from the variation with time of the oxygen concentration in the exhaust gases. The signal delivered by the $O_2$ analyser has to be corrected to take account of water condensation in a cooled trap.

The weight loss of the sample is measured by an electronic balance and the effective heat of combustion is calculated by:

$$HC = \frac{(QT_1 + QT_2)}{m_b} \quad \text{or} \quad HC = \frac{QO_2}{m_b} \quad (m_b : \text{ mass burning rate , kg.s}^{-1}).$$

Air or air enriched by oxygen or nitrogen flows around the sample at a controlled overall flowrate. The surface of the sample can be heated by a radiative flux emitted by four quartz lamps. Presently, the maximum heat flux is close to $3.5 \text{W.cm}^{-2}$ with changes with the position at the surface lower than 10%. Improvement in the efficiency of the lamps is under study to increase the maximum incident heat flux.

RESULTS

Methane-air flame
The objective of the experiments performed with a cylindrical gas burner was both to calibrate the calorimeter and to control the inertia of the thermal measurements. The second point is important, especially for composite materials that can lead to an evolution of the effective heat of combustion due to staged thermal degradation of the solid.
The very first experiments were conducted with a constant methane volumetric flowrate. Both the thermal and the oxygen consumption method gave effective heat of combustion in close agreement with the literature value (50 kJ.g\(^{-1}\)).

In a second step, the effect of step by step variations in the methane flow rate has been assessed. Temperature measurements and oxygen concentration evolution for a 4 steps experiment with methane flowrate adjusted to 12.6 cm\(^3\).s\(^{-1}\) (from 500 to 1500 s), 26.2 cm\(^3\).s\(^{-1}\) (1500 to 2500 s), 51.0 cm\(^3\).s\(^{-1}\) (2500 to 3500 s), 26.2 cm\(^3\).s\(^{-1}\) (3500 to 4500 s), have been plotted on figure 1. Corrections of the signal with the Evans and Breden's technique (4) lead to time response close to 60 s for both measurements. Hence thermal inertia is considerably smaller than those reported for ISO (666 s) and ASTM (200 s) calorimeters (5). Figure 2 shows that a constant value is obtained for the effective heat of combustion in these experiments.

Combustion of PMMA
The effective heat of combustion of PMMA has been studied in air, without additional heat flux. Evolution of the mass burning rate shows an increase up to a maximum, due to an increase in the area of the combustion zone, followed by a continuous decrease corresponding to the consumption of the sample.

In spite of this non stationary character, a constant value of the effective heat of combustion is obtained from measurements performed during 90% of the whole experiment. Moreover this value (24 kJ.g\(^{-1}\)) is in close agreement with literature values for this material: 23 - 27 kJ.g\(^{-1}\) (6,7).

Combustion of PVC
Experiments on PVC have been conducted with oxygen enriched air (37% O\(_2\)) and an external radiant heat flux (2.8 W.cm\(^{-2}\)). The mass burning rate and the oxygen consumption curves exhibit a two stages evolution. Both measurement techniques lead to a slight increase in the effective heat of combustion (15 to 18 kJ.g\(^{-1}\)).

REFERENCES
HOT SURFACE IGNITIONS: IONIC EFFECTS IN IGNITION INITIATION

William W. Bannister, Department of Chemistry
University of Massachusetts, Lowell, MA 01854
[508] 934-3682

Douglas F. Nelson, Air Force Civil Engineering Support
Agency [RACF], Tyndall AFB, FL 32403-6001

ABSTRACT

Events preceding spontaneous ignition of fuels on hot surfaces may be ionic rather than free radical, with polarizing hot surface effects and anionic stabilizing influences within the fuel molecules. Adsorbed oxygen and fuel molecules may be thermally polarized at the surface, with the negative end of the oxygen dipole abstracting a proton from the fuel molecule to form a carbanion/hydroperoxy cation pair. Simple electron transfer mechanisms between carbanionic and hydroperoxy cationic moieties would then form free radical species, these then participating in conventional cool and hot flame free radical mechanistic pathways. Structure/property correlations for ignition temperatures for various hydrocarbon fuel molecules are consistent with carbanionic mechanisms, and inconsistent with effects typically associated with stabilizations of free radicals. Inclusion of polar compounds (e.g., water, hydrogen bromide and bromofluorocarbons) into fuel/air mixtures facilitates spontaneous ignition. Ignition temperature trends for
metallic surfaces are consistent with electron work functions (these being proportional to ease of electron migration from the metal surface): metals having lower work function values have lower ignition temperatures than those with higher work function values. Highly polar oxyanionic surfaces such as quartz greatly facilitate hot surface ignitions. These facts are all consistent with ionic rather than free radical effects in pre-ignition stages.

An inverse relationship exists between molecular weight and ignition temperatures: up to a point, volatile higher molecular weight fuels have lower ignition temperatures and are more easily ignited by hot surfaces. Branched and cyclic alkanes, arenes and olefins have higher ignition temperature than analogous straight chain alkanes. Important structural parameters in governing ignition temperatures include: effects of molecular weight on the velocity of the fuel molecule near the hot surface; effects of rigidity of the fuel molecule on its ability to recoil from the hot surface; and effects of specific heat of the molecule with regard to ability for dissipation of heat energy within the molecule.
EXPERIMENTAL STUDY OF SELF-IGNITION OF ELECTRICAL CABLES EXPOSED TO AN EXTERNAL RADIANT FLUX
A Simplified theory for generalizing results

Mane-Claude Lopez, Jaques Bergelin
Institut de Protection et de Surete Nucleaire
Departement de Protection de l'Environnement et des Installations
Service d'Etudes et de Recherches en Aerocontamination et en Confinement

We present an experimental procedure to get some quantitative data on the pyrolysing and burning characteristics of combustible materials which may be found in nuclear facilities. In this study, the materials considered are only polymeric cable insulation and jacket which are flammable solids. In order to accomplish this goal, the experimental installation allows the testing of one to twelve cables. The cables are 50 cm long. They are fixed vertically on a metallic tray and exposed to irradiance levels ranging from 2 to 6 $Wcm^{-2}$.

The test is performed in a realistic fire situation i.e. unconfined normal air environment, without piloted ignition. The results presented here concern one type of cable i.e. a 7 copper conductor cores, PVC jacket and PRC insulation. A schematic of the experimental set up is shown figure 1.

The apparatus consists basically of a gas fired radiant panel (30x30 cm) which imposes a known flux distribution over the sample. The sample holder frame, where the cables are vertically fastened with steel tie wire, is put on a load cell for mass loss data collection. An insulating sliding shutter is located between the radiant panel and the specimen to precisely control exposure times.

The pyrolysis and combustion products are collected in a hood for analysis. Cable temperatures and exhaust gas temperatures are measured by Type K chromel-alumel thermocouples. For calculating the rate of heat release, the temperature, oxygen concentration and flow rate of the exhaust gases are to be measured. Oxygen consumption is performed by a continuous sampling paramagnetic oxygen analyzer. Exhaust gas flow rate is measured in the exhaust duct by the helium gas tracer technique. The measurements of the products of combustion such as carbon monoxide, carbon dioxide, unburnt hydrocarbons are carried out by a continuous sampling infrared analyzer. Data reported typically include ignition delay time, mass release rate at various flux levels, heat release rate and products of combustion. The experimental results show a linear relationship between the mass release rate ($m^{\prime}\text{stat}L^{-1}\text{cm}^{-2}$) for a stationary state and net heat flux delivered to the cable surface $Q_r$ in $Wcm^{-2}$ for a given number of cables. The inverse of the slope of the straight gives us the heat of gasification $L_g (Jg^{-1})$.

$$m^{\prime}\text{stat}=\frac{1}{L_g}(Q_r-Q_c), \text{ relationship proposed by Tewarson and Pion.}$$

In this relationship, $Q_c$ is the heat loss from the surface ($Wcm^{-2}$). The Observation of this phenomena led us to the definition of a Critical exposure heat flux $Q_{rc}$ which can be linked to the number of cables.

<table>
<thead>
<tr>
<th>Number of cables</th>
<th>Combustible mass in g</th>
<th>Critical heat flux in $Wcm^{-2}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 cable</td>
<td>77</td>
<td>4,95</td>
</tr>
<tr>
<td>2 cables</td>
<td>155</td>
<td>4,75</td>
</tr>
<tr>
<td>4 cables</td>
<td>310</td>
<td>4,6</td>
</tr>
<tr>
<td>8 cables</td>
<td>642</td>
<td>3,6</td>
</tr>
<tr>
<td>12 cables</td>
<td>930</td>
<td>3</td>
</tr>
</tbody>
</table>
In our work, we try to confirm the approximate expression (Tewarson and Kahn) which deduced the ignition delay time from the sample surface temperature and the external radiant flux. This type of relationships between the exposure conditions and material behavior should prove useful in attempts to predict the development of fires using deterministic computer codes.

References:


ARGOS
A computer program for fire risk evaluation

Niels Baden
Danish Institute of Fire Technology,
Datavej 48, DK-3460 Birkeroed, Denmark

A new and user friendly software package **ARGOS has been** developed for fire prediction. The computer program is fully menu driven with full on-line, context sensitive help facility and simulation speed second to none due to **high** efficiency numerical methods. The central module in ARGOS is a zone model for simulation of fire development and smoke spread in buildings including:

- Simultaneous calculation of up to five compartments.
- Unique physical models for fire growth and burnout in combustible objects.
- Fuel/ventilation controlled fires.
- Smoke development and spread.
- Temperature development in compartments and smoke layers.
- Temperature development/heat penetration in building components.
- Automatic fire alarm, sprinkling and fire ventilation.
- Estimation of damage to buildings, stock and machinery caused by the fire in question.

Database with built-in or user specified background data for simulation of fire development, including:

- Building components such as doors, windows, ceilings etc.
- Fire start, e.g. a three meter high stack of pallets.

**ARGOS V1.0** Copyright (C) 1990, 1991 Danish Institute of Fire Technology

- Main menu
  - 1 Client files
  - 2 Calculate
  - 3 Database
  - 4 Database
    - 1 Basic building constr.
    - 2 Fires
      - 1 Solid material
      - 2 Solid material
      - 3 Create new
      - 4 Display/edit
      - 5 3 Delete
      - 6 4 Print
      - 0 Help

- Fires in solid materials/edit
  - Name: [Sofa, 3 persons]
    - Horizontal fire spread [m/min]: 0.70
    - Max. heat release rate [MW/m³]: 1.75
    - Local burn-out time [min]: 1.80
    - Initial flame height [m]: 1.00
    - Vertical doubling time [min]: 1.00
    - Height of stock [m]: 0.70
    - Width of stock [m]: 0.70
    - Length of stock [m]: 2.10
    - Optical smoke potential [dB/m]: 400.0

- F1-Topics PgUp/PgDn-Page t1-Line Esc-Previous

*Database: Window for specification of parameters for fire start model.*
ARGOS has been tested against three well documented full scale test series:

- The two rooms tests series reported by Cooper et al. (1982). These tests are often referred to for model verification, though the rooms and fires studied are relatively small. 
- The two rooms test series including fire ventilation in the roof reported by Hägglund (1992). In these tests the rooms and fires are relatively large.
- The three rooms test series reported by Meland & Lønvik (1989). These tests also include various fire detectors.

The comparisons with full scale fire tests shows that the basic principles and implementation of ARGOS leads to satisfactory simulation of a wide variety of fire scenarios.

References


ANALYSIS OF THE NATIONAL FIRE PROTECTION ASSOCIATION
FIRE RISK ASSESSMENT FRAMEWORK: EXAMPLE CALCULATION

F. Merrill Galloway¹ and Marcelo M. Hirschler²
¹ BF Goodrich Technical Center
P.O. Box 122
Avon Lake, OH, 44012
² Safety Engineering Laboratories
38 Oak Road
Rocky River, OH, 44116

The National Fire Protection Association Research Foundation completed a project to develop a Fire Risk Assessment Methodology (FRAM). In order to check the validity of the software developed, the technical team used it to determine fire risk in four cases: upholstered furniture in residential environments, carpets in offices, concealed combustibles in hotels and motels and wall linings in restaurants. This project was a great technical success, since FRAM was capable of adequately predicting fire risk in, at least, the first three applications attempted.

One of the main objectives of the development of FRAM, however, was to allow users to compare the fire risk associated with new products with that associated with the traditional products in use today.

This work involves just such a comparison, for a sub-case of the first example: flaming upholstered furniture in a residential ranch house. A guide to the FRAM methodology software, accompanied by this example case, was supplied to the authors by NIST. The objective of supplying this was to carry out "beta testing", in order to confirm that the same results could be obtained by different investigators. The objective was achieved since the results from NIST were confirmed, except for a minor glitch: in the results of the beta test the risk appeared greater with smoke detectors than in their absence. This was clearly a case of one misaligned command, and is of no real consequence.

The present work involved further analysis however. A set of eight additional cases were run, in which the inputs were changed. The changes involved the heat of combustion of the fuel (the upholstered chair), the time intervals specified for the fire, the mass loss rate in each time interval and the area of the fire. In 7 of the cases the heat of combustion of the fuel was decreased from 14 MJ/kg (base case) to 9 MJ/kg and in the other case (# 5) it was raised to 35 MJ/kg. Table 1 shows the fire risk results, which show almost no difference between all the cases investigated, which cover a period of 760 s.

In view of the disappointing results, a few additional tests were run with the same data. Firstly, it was found that, although the overall fire risk result did not change much in the various cases, the individual causes of death changed considerably. For example in the base case the major cause of death was # 5 (FED Purser, involving heat and toxicity), while in case 5 (with high heat of combustion) the major cause of death was # 2 (temperature). The major cause of death in all the cases with lower heat of combustion was also # 5.
Figure 1 shows a comparison of upper layer temperatures in three cases (1, 4, 5) which are identical in every respect except for the heat of combustion of the fuel. It is clear that the temperatures are very different, both in the final value and in the time required to reach that value. In fact, in case 4 flash temperatures are not even reached in the time period under consideration. This is not, at least on its face, compatible with virtually no difference in overall risk.

The results of this investigation indicate that the FRAM requires a very careful analysis before attempting to compare the fire risk of a new product with those of an existing product. The differences in fire hazard (for example from toxicity or temperature) between the two cases are very clearly noticeable, but they do not always translate easily into fire risk differences.
AN EXPERIMENTAL INVESTIGATION OF THE PULSATION FREQUENCY OF FLAMES

A. Hamins, J.C. Yang and T. Kashiwagi
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

A large number of experimental, theoretical and scaling studies have investigated the coherent vortical structures which are shed by flames [1]. This phenomenon has been documented for a wide range of burner diameters, heat release rates, Reynolds numbers and fuels. The vortical structures and their shedding frequency can influence the rate of air entrainment into a fire. As the pulsation frequency affects the flow field, it may also affect the flame length, combustion efficiency, and flame radiance.

In the fire literature, the pulsation frequency \( f \) has been typically correlated with the burner diameter \( D \) which yields a power law fit to the data such that \( f \propto D^{b.5} \). Implicit in this correlation is the idea that the pulsation frequency is not a function of fuel exit velocity at the burner, fire heat release rate or other parameters.

Measurements of the pulsation frequency in non-premixed flames were conducted for gaseous and liquid fuels. Measurements were performed over a wide range of Froude number \((10^{-4} \text{ to } 10^3)\), Reynolds number \((10 \text{ to } 10^3)\) and burner diameters \((0.0074 \text{ m to } 0.30 \text{ m})\). A video camera \((30 \text{ frames/s})\) was used to record the flame image for several minutes. The characteristic pulsation frequency was determined by counting the number of frames necessary for the vortex growth process to cycle. The standard deviation of the frequency measurement was approximately \(0.2 \text{ Hz}\).

The fuel velocity at the burner exit was found to have a weak influence on the pulsation frequency for some diameters whereas total heat release did not. Figure 1 shows that a plot of the Strouhal number as a function of the inverse Froude number correlates the measurements made here as well as measurements reported in the literature for pulsations in flames burning gaseous, liquid and solid fuels over 14 orders of magnitude in Froude number, ranging from buoyancy dominated to momentum dominated flames. Here the Strouhal number \( S \) is a non-dimensional frequency defined as:

\[
S = \frac{f \cdot D}{V}
\]

where \( V \) is the average velocity at the burner exit. The Froude number \( (Fr) \) is defined as the ratio of inertial to buoyant forces:

\[
Fr = \frac{V^2}{(D \cdot n \cdot g)}
\]

where \( n \) is the acceleration normalized by \( g \) \((9.8 \text{ m/s}^2)\), the earth normal gravitational acceleration. Previous measurements \([2]\) of the effect of enhanced gravitational level on the pulsation frequency were also interpreted in terms of the Strouhal-Froude number relationship as shown in Fig. 1.
A shadowgraph technique was used to measure the pulsation frequency of an isothermal buoyant gas stream over a range of Froude number (10⁻³ to 1). Figure 1 also shows that the Strouhal number as inverse Froude number plot correlates the measurements, but yields a different power law exponent than the reacting flow case.

Stability limits in flames were also investigated. The critical fuel velocity (Vc) needed to initiate pulsations as a function of burner diameter for methane and propane flames was found to be: Vc ∝ D⁻¹.5.

References


ACQUISITION AND ANALYSIS SYSTEM OF BURN INJURIES
FROM FLAMES AND THERMAL RADIATION

Barry S. DeCristofano, Gerald J. Caldarella,
Brian R. Kimball, and Joseph F. Roach

U.S. Army Natick R&E Center
Natick, MA 01760

An advanced thermal response data acquisition system has been custom designed and built by Natick to provide state-of-the-art
collection and analysis of the extent of burns induced by both flames and thermal energy transport through clothing systems. Data
can be recorded in a laboratory setting or sent for miles on a single coaxial cable, fiber optic cable or FM frequency. A total of
521 sensors (124 on each of four manikins and 25 variably located) provide a temperature-versus-time record of the temperatures reached
in the skin tissue of a hypothetical wearer of the clothing system.

Using software tailored to the system, the researcher acquires data to be displayed in real time or stored for analysis of burn-severity
predictions at any convenient future time. The software used in this process has been written by Natick specifically for thermal response
investigations. The data recorded from the experiment are used to generate two- and three-dimensional representations of the extent of
skin burns predicted from skin simulant recordings. The
three-dimensional manikin display is capable of showing the
progression of skin damage as it changes with time. The software
can be linked to models used by clothing and equipment designers to allow projection of these articles onto the image. The four-manikin
system can be exposed in flame-pits, vehicle crew-compartment, and
structures. This advancement of the technology used by the Army for over 20 years allows for more rapid analysis and greater statistical
confidence in the data obtained in thermal-injury studies. More complete utilization of the areal extent of thermal sources is also
achieved by the use of multiple manikins. This system is a complete package, which gives researchers the tools required to study
accurately and comprehensively the effects of flame and thermal radiation on wearers of military and/or civilian clothing.
INTERRELATIONSHIP BETWEEN HYDROGEN CYANIDE GAS EXPOSURE CONCENTRATION, EXPOSURE TIME AND BLOOD CYANIDE LEVEL IN RATS

Arvind K. Chaturvedi, Boyd R. Endecott, Roxane M. Ritter and Donald C. Sanders

Biochemistry Research Section, Toxicology and Accident Research Laboratory, Civil Aeromedical Institute, Federal Aviation Administration, Post Office Box 25082 (AAM-613), Oklahoma City, Oklahoma 73125-5066

Inhalation of toxic combustion gases is a significant cause of death in aircraft fires, and hydrogen cyanide (HCN) gas is one of the principal toxicants generated in potentially lethal amounts during fires (Gad, 1990; Baud et al, 1991; Mayes, 1991; Veronneau et al, 1992). This gas has the ability to produce incapacitation at relatively low concentrations (Crane et al, 1989; Hartzell 1989; Gad, 1990), and time-to-incapacitation ($t_i$) is an applicable toxicological index for predicting escape from a fire (Crane et al, 1977; Spurgeon et al, 1979; Sanders et al, 1991). In HCN poisonings, blood cyanide (CN⁻) levels are frequently measured to explain the severity of HCN gas exposure; how blood CN⁻ levels are related to HCN exposure concentrations and exposure times is not well defined. Therefore, this study on the HCN gas uptake was conducted to establish a possible interrelationship between HCN gas concentration, exposure time and blood CN⁻ level; the uptake was measured as increases in blood CN⁻ levels as a function of exposure time for 184 and 64 ppm HCN that produce nominal 5- and 35-min $t_i$ in rats, respectively. These HCN concentrations were selected to further establish the applicability of the interrelationship at these $t_i$ periods, representing both evacuation and in-flight-plus-evacuation periods (Crane, 1984, 1989; EUROCAE, 1991).

Male Sprague-Dawley rats (100-125 g) were individually exposed to HCN gas in a chamber equipped with a cage rotating at a circumferential velocity of 8.5 cm/sec. The exposure times were 1, 2, 3 and 4 min (4 rats/exposure interval) for the 5-min study, while they were 2.5, 5, 10, 15 and 25 min for the 35-min study (3 animals/exposure interval). At the end of each exposure interval, rats were removed from the chamber and killed by cervical dislocation for the blood collection and CN⁻ determination. Average HCN concentrations for each experiment were obtained by the integration of chamber HCN concentration ($C_\text{ch}$; ppm) as a function of exposure time ($t$; min) and dividing the resulting $C_\text{ch}$t product by $t$. Chamber HCN concentrations and blood CN⁻ levels were measured colorimetrically using a modification of the Technicon Industrial Method No. 312-74W (Technicon, 1974); this method includes the conversion of complexed CN⁻ to HCN by the digestive distillation of samples at 155°C and resampling the HCN condensate. Data were evaluated at a $= 0.05$ using the analysis of variance, Tukey's HSD multiple comparison test and Student's $t$-test; values are presented as the mean ± SD.

HCN concentrations for the uptake studies were 183 ± 4.4 (n = 16) and 71 ± 3.4 (n = 15) ppm. Within each set of studies, HCN concentrations for the exposure intervals were not different from each other ($p > 0.05$). The HCN gas uptakes at the two concentrations as represented by blood CN⁻ levels vs exposure times were linear; there was no indication of a plateau effect prior to 5 min for 183 ppm HCN or 35 min for 71 ppm HCN. Slopes indicated that blood CN⁻ increased at 0.401 and 0.152 µg/mL/min for 183 and 71 ppm HCN, respectively. There was a direct relationship between the HCN chamber concentration and uptake rate, as the decrease in the HCN concentration by a factor of 2.6 also decreased the blood CN⁻ uptake rate by 2.6. Calculated uptakes in (µg CN⁻ /mL)/min/ppm HCN for both exposures were almost identical, i.e., 2.19 x 10⁻³ for 183 ppm HCN and 2.14 x 10⁻³ for 71 ppm HCN. Thus, this relationship can be described by

$$\frac{(CN^-)}{C \cdot t} = K$$

which can be rearranged to the form
\((\text{CN}^-) = C \cdot t \cdot K\)

where \(\text{CN}^-\) = blood \(\text{CN}^-\) in \(\mu\text{g/mL}\); \(C\) = HCN chamber concentration in ppm; \(t\) = exposure time in min; \(K\) (constant) = \(2.2 \times 10^{-3}\). This equation may have some utility for predicting blood \(\text{CN}^-\), HCN exposure concentration, or exposure time value by knowing the values of two of the three parameters (variables).

In a separate study, where \(t_i\) was measured as time from the insertion of rat until it could no longer walk in the rotating cage, blood \(\text{CN}^-\) levels of 2.3 and 4.2 \(\mu\text{g/mL}\) were observed at incapacitation from exposures to 184 and 64 ppm HCN, respectively, which corresponded to values of 2.1 and 4.4 \(\mu\text{g/mL}\) calculated from Equation (2). When \(K\) was derived from parameters obtained in the 184- and 64-ppm HCN experiments, a mean value of \((2.3 \pm 0.5) \times 10^{-3}\) was obtained \((n = 100; t = t_i)\). This value is very close to the \(K\) value calculated from the uptake study, suggesting that Equation (1) holds true for incapacitation, as well. Therefore, blood \(\text{CN}^-\), HCN concentration, or \(t\) (or \(t_i\)) can be predicted from the equation. However, there was no specific blood \(\text{CN}^-\) level that could be linked to the onset of incapacitation.


Performance of the Reciprocal Nephelometer

GW. Mulholland N.P. Bryner

Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg, MD 20899

The optical properties of combustion generated aerosols, which we shall refer to as smoke, are of concern in regard to the degradation of visibility in building fires as well as in pristine areas, to the global climatic impact of a major nuclear exchange as well as the localized climate impact of forest fires and the oil well fires such as those recently extinguished in Kuwait. There is a large uncertainty in the optical properties of smoke including the scattering and absorption coefficients per unit concentration of smoke and the single scattering albedo defined as the ratio of the total scattering coefficient to the extinction coefficient. A comparison study involving simultaneous measurements of the optical properties of smoke generated from the combustion of propane resulted in values ranging from 3.8 to 11.4 m$^2$/g for the absorption coefficient and 0.09 to 0.29 for the single scattering albedo.

Gerber developed an instrument for simultaneously measuring the light extinction coefficient $\sigma_{ext}$ and the total scattering coefficient, $\sigma_s$, of ambient atmospheric aerosols with a multi-pass optical cell. From these measurements, the single scattering albedo is obtained as the ratio, $\sigma_s/\sigma_{ext}$, and the light absorption coefficients is obtained as the difference, $\sigma_{ext} - \sigma_s$. The total scattering measurement in Gerber's instrument utilizes a diffuser before the detector and is the inverse of traditional nephelometry, which uses a diffuser to provide a cosine distribution of the light source. We use the term transmission cell-reciprocal nephelometer (TCRN) to describe this instrument. Patterson et al. developed a single pass TCRN and used it to characterize the optical properties of smoke generated by the combustion of lumber, various plastics, petroleum products, and tire rubber.

This study assesses the accuracy of the TCRN for the total scattering measurement through a combination of modeling and measurement. The model calculations consist of computing the total flux of light scattered from particles in the laser beam to the cosine sensor/detector in the TCRN. The inference of the total scattering coefficient from the detector output is based on a number of assumptions: The area of the detector is infinitesimal, the diffuser has a perfect cosine response, and the acceptance angle extends from 0° to 180°. In this paper we compute the effect of each of these assumptions on the accuracy of the inferred result. These effects are computed for monodisperse polystyrene spheres over the size range 0.02 - 8 μm based on Mie theory and for smoke agglomerates ranging from 10 primary units to $10^7$ primary units based on the Fisher-Burford form for the scattering function. The accuracy of the model calculations is determined by comparison with exact solutions for the case of a detector with an infinitesimal area and for a finite area detector with a diffuse scattering function.

The predicted results are compared with measured results for six different sizes of monodisperse polystyrene sphere aerosols with particle diameters in the range 0.1 to 235 μm. The measurements were carried out as a function of the distance between the laser beam and detector for a 13 cm and a 2.7 cm diameter cosine sensor. Based on these measurements, our TCRN design with a 1.3 cm cosine sensor positioned 24 cm from the detector ($\theta_{min}$=2.4°) has an accuracy for $\sigma_s$ of about...
5% for spherical particles with diameters up to 1.1 μm. We estimate similar accuracy for smoke agglomerates with up to $3 \times 10^3$ primary spheres. A table of design parameters for making accurate total scattering measurements are obtained for both spheres and agglomerates.

References


CORRELATIONS FOR TEMPERATURE AND VELOCITY IN ROOM FIRES FOR ESTIMATING SPRINKLER ACTIVATION TIMES

by

PRAVINRAY D. GANDHI, SENIOR STAFF ENGINEER
UNDERWRITERS LABORATORIES INC.

ABSTRACT

There have been significant developments in the understanding of the fire initiation, fire growth, and suppression mechanisms. The understanding has helped the development of predictive methods that estimate ignition of a product assembly: the subsequent fire growth, and also estimate fire hazards. Further, methods are also available to predict, the activation time for automatic sprinklers, if present. These predictive methods are in the form of engineering correlations that provide relatively quick estimates of fire hazards [1], and in the form of computer fire models [2] that predict the fire initiation, its growth and fire hazards for a specific fire scenario considered.

Several engineering correlations have been used to develop computer fire models. One such model is DETACT-QS [3] in which the correlations of temperature and velocity have been used to predict the activation time for sprinklers located under unconfined flat ceilings. This model may also be used for predicting activation times in large rooms where the influence of confining walls on temperature and velocity of the hot gases are negligible. However, the use of DETACT-QS may not be applicable to many residential occupancies since these occupancies have relatively smaller rooms and the influence of confining walls on the temperature and velocity of hot gases is expected to be significant. Further, room ventilation is also expected to have a significant influence on the temperature and velocity of the hot gases. Correlations that may be applicable for these type of occupancies are not currently available.

In this paper, room fire tests conducted to develop temperature and velocity data are described. The tests were conducted in rooms ranging in size from 2.44 m x 2.44 m x 2.44 m high to 4.27 m x 4.27 m x 2.44 m high. A single or a double doorway was used for ventilation. Soldered and glass bulb sprinklers were located at the ceiling (pendant style) and the wall (sidewall style). A sand burner located in the center of the room was used as the fire source to provide three fire sizes (73 kW, 100 kW, and 147 kW). The room was instrumented to measure the temperature and velocity at specific locations, and monitor the activation of sprinklers.

The temperature and velocity data were analyzed and it was found that both the temperature increase and the velocity of the gases at the sidewall style sprinkler location were significantly lower than observed at the pendant style sprinkler location. This is expected to be due to higher heat and friction losses for the sidewall location. Further, the test data showed that the time for the hot gases to attain steady state temperatures was significant in relation to the sprinkler activation time.

The temperature and velocity data were then further analyzed and used to develop nondimensional correlations for temperature, and velocity for both the pendant and sidewall sprinklers. Additionally, the quasi-steady assumption used in some sprinkler activation models was examined to determine its applicability for residential sprinklers, and a nondimensional correlation for the temperature transient observed in the hot gas layer during the fire tests was developed.

The nondimensional correlations were of the form

\[ \Gamma = \kappa \left( \frac{Q}{\sqrt{\rho_0 C_{p0} T_0 A_0 H_0}} \right)^{\frac{1}{2}} \left( \frac{h_k A_k}{\sqrt{\rho_0 C_{p0} A_0 H_0}} \right)^{\frac{1}{2}} \]

where \( Q \) is the heat release rate, \( g \) is acceleration due to gravity, \( \rho_0 \) is density of air at ambient conditions, \( C_{p0} \) is specific heat of air at ambient, \( T_0 \) is ambient temperature, \( A_0 \) is the area of the vent, \( A_k \) is the height of the doorway, \( h_k \) is the heat transfer coefficient for heat loss through the walls of the test room, and \( A_k \) is the total surface area of the room (walls and ceiling). The factor \( \Gamma \) was defined as (i) \( \Delta T_g / T_0 \) - nondimensional temperature increase (\( \Delta T_g \) is steady state temperature increase); (ii) \( U / (\sqrt{\rho_0 A_0 H_0 / A_k}) \) - nondimensional velocity (\( U \) is velocity); and (iii) \( \alpha / (\sqrt{\rho_0 A_0 H_0 / A_k})^{3/2} \) - nondimensional temperature transient factor (\( \alpha \) is a time constant for temperature increase in the room). The proportionality constant \( \kappa \), and the
exponents M. and N were determined using the method of least squares for both the pendant style and sidewall style sprinklers from the data obtained from the series of fire tests.

The resulting correlation for temperature for pendant style sprinklers is in reasonable agreement with existing correlation developed by McCaffery [4] for room fires. The new correlation for velocity provides closer agreement to test data than current correlations. The results show that assuming quasi-steady conditions in the room will underestimate the time for sprinkler activation. The paper demonstrates that the use of transient temperature factor developed allows for a closer correlation between the actual activation time and the predicted actuation time.

REFERENCES:


SPRINKLER HEAD RESPONSE MEASURED AND PREDICTED

Haukur Ingason
Swedish National Testing and Research Institute
Bords, Sweden

Thermal response models for glass bulb sprinklers have been studied considering the following thermal response parameters; the Rate of time Index (RTI), the Conduction parameter (C), and the Change of Phase parameter (CHP). The RTI parameter reflects the thermal time constant of the glass bulb and the C parameter the heat conduction loss to the sprinkler fitting. The CHP parameter is thought to be a factor accounting for a time delay shortly before activation caused by heat absorption of the operating medium or work needed to shatter the glass bulb.

The use of three parameters in the response model instead of two, i.e. RTI and C, have been proposed to describe the thermal response of both glass bulb sprinklers and solder type sprinklers. According to the three-parameter model, the heat-sensitive element is heated by convection from the air stream and cooled by conduction to the sprinkler supports up to a point where the element starts to absorb heat energy without a corresponding rise in temperature. CHP characterizes, therefore, a physical sequence, starting when the element temperature has reached a value close to the activation temperature. For solder type sprinklers the absorbed energy is assumed to arise from the heat of fusion.

Combinations of one, two and three parameters have been used to predict response times for various plunge and ramp test conditions, as well as for a wide range of growing fire conditions. These calculations indicate that the test results cannot be fully explained in the framework of the thermal response models applied. Firstly, consistently different C values were obtained for ramp tests with different rates of temperature rise. Secondly, tests with precondition temperature of the sprinkler close to the operation temperature resulted in higher RTI values compared with cases with normal precondition temperatures. A thermal analysis with the finite element computer program SUPER-TASEF shows that a plausible reason for the different C values is uneven heating of the sprinkler frame arms during different ramp tests while the higher RTI values can be attributed to a time delay caused by temperature gradients within the glass bulb. This indicates that the CHP parameter found necessary to explain some earlier test data in terms of heat absorption or work needed to shatter the glass bulb, more likely reflects an effect of thermal time delay.

Thermal analysis showed that the time delay will yield different RTI values in a plunge test for different heating conditions and consequently the glass bulbs will not respond as predicted. It was also found that the relative effects of the time delay on the RTI value will increase with shorter response times. However, the time delay is not constant for all types of heating conditions. The time delay was found relatively constant for the plunge test conditions, while for time varying heating conditions, such as those in ramp tests, no time delay was found. This has been indicated earlier by Job et al. Calculated heat release rate at sprinkler response in a growing fire condition was used to evaluate the thermal response models. For the fast, medium and slow fire growth rates the heat release rates at sprinkler response were found to be similar, whether based on calculations with two parameters (RTI and C) or with all three parameters (RTI, C, CHP). However, when using only the RTI value, the predicted rate of heat release at sprinkler response decreased significantly.

The velocity around the frame of one of the tested sprinklers has been measured at different orientations using Laser Doppler Velocimetry (LDV). The objective was to measure the air flow conditions around the sprinkler frame arms at two different orientations. A field model simulation
was carried out for same type of sprinkler to enable a comparison with the measured values. The comparison between measured and calculated velocities show good agreement.

When the sprinkler frame arms are orientated parallel to the direction of air flow it is not obvious that the Nusselt number should correlate to the square root of the Reynolds number, as it does when the frame arms are perpendicular to the air flow. One of the basic assumptions in the RTI theory is that the heat transfer to the heat-sensitive element correlates to the square root of the Reynolds number and thus the square root of the air flow velocity. The heat transfer to the glass bulb is proportional to the Nusselt number. When the sprinkler frame arms are orientated parallel to the air flow, the flow conditions around the glass bulb will change due to the shadowing effect of the frame arms. This will alter the convective heat flow to the glass bulb compared to the case where the frame arms do not obstruct the flow. To investigate the correlation between the Nusselt number and the Reynolds number for the glass bulb when the frame arms are orientated parallel to the air flow indicate that the heat transfer varies with the square root of the velocity as assumed in the original RTI theory. The RTI theory is therefore deemed to be valid for the simulated sprinkler irrespective of the orientation.

References


Extended Abstract

This paper presents the results of a study performed to determine the thermal radiation from the unexposed face of various classes of insulated bulkheads with one face exposed to a prescribed furnace exposure. The bulkheads were representative of that which may be found in marine vessels. Full scale Class A-60, A-30, A-15 and A-0 bulkheads were evaluated in accordance with Fire Test Procedures for "A", "B" and "F" Class Divisions (Res.A.517(13)) published by the International Maritime Organization. The thermal radiation and surface temperature measurements have defined the range of levels to which passengers would be exposed if the need for egress along a fire boundary became necessary. These results have shown that all classes evaluated to date (A-60, A-30 and A-15) other than the A-0 class should provide adequate protection, i.e., safe passage through a corridor. In addition, the information obtained forms a useful database for the thermal behavior of the materials used in the evaluations. In addition the results have fit expected scaling behavior based on dimensional analyses. Finite-element transient heat transfer modeling was performed to predict the effect of density and thickness of insulation on the thermal failure time of the bulkheads with a high degree of success, as shown by comparison with the actual data.

A Class "A" fire boundary is a bulkhead which is constructed of steel or other equivalent material, is suitably stiffened, and is so constructed as to be capable of preventing the passage of smoke and flame to the end of a one-hour duration of the test method: Recommendation on Fire Test Procedures for "A", "B", and "F" Class Divisions (IMO Res. A.517(13)) as it applies to Class "A" bulkheads. Further refinement of the classification is based on the time period for which the rise of temperature of the unexposed side is limited. In particular, an uninsulated steel bulkhead shall be Class A-0, while other classes shall be insulated with an approved structural insulation. The average temperature of the unexposed side will be less than 139°C above the initial temperature without rising, at any one point, including any joint, more than 180°C above the original temperature at the end of 15 minutes for a Class A-15; 30 minutes for a Class A-30; and 60 minutes for a Class A-60.

To briefly describe the setup and evaluation method, the bulkheads were constructed of 4.5 mm thick mild steel, 2.4 m high by 1.9 m wide. Vertical stiffeners, 65 by 65 mm, were spaced 600 mm on center. Several configurations were evaluated, depending on the classification desired. Evaluations were performed with no insulation (A-0), with mineral wool (A-60, A-30, A-15), or a combination of mineral wool and calcium silicate marine board (A-30). The construction was typical of that found on marine vessels. Also, the A-60 consisted of two configurations, one with the insulation exposed to the fire, and the other with the steel exposed to the fire. A minimum of three evaluations per configuration were performed, to provide for repeatability, as well as to allow the results to encompass the
expected performance rating. In general, the density or thickness was varied slightly so that the actual rating would bracket the "ideal" rating. The evaluations were performed using SwRI's vertical furnace, under positive pressure. The furnace was calibrated to assure a uniform exposure over the surface of the bulkhead. The furnace temperature curve was maintained in accordance with the IMO test standard, which is generally a logarithmic fit reaching 945°C at the end of a one hour period.

The data obtained included thermal radiative measurements of the face of the bulkhead, surface temperatures, furnace temperatures, and for some of the evaluations, thermography (infrared thermal imaging) was used to view the entire unexposed surface. The most significant results obtained from these evaluations has been in the development of a database of unexposed surface temperature and heat flux for Class A-60, A-30, A-15, and A-0 bulkheads. This data contains information very useful for fully characterizing the behavior of insulated and uninsulated bulkhead systems. This information can be used to predict the actual expected levels of radiation in the event of fire, and also to determine regulatory guidelines for the use of these materials and alternative materials. As seen in the literature, the thermal radiation measured brackets the range of acceptable exposure levels, depending on the class and configuration of bulkhead evaluated.

The data obtained has been submitted to dimensional analyses, with a high degree of correlation. This will be very useful for the prediction and modeling of fires in marine vessel construction. The analysis has shown that the primary factors affecting the thermal protection offered by mineral wool are the density and thickness of material used. Further study of the scaling parameters may broaden the scope of applicability of the correlations obtained.

The thermal modeling performed has been very useful in selecting the density and thickness of insulation to meet the requirements of the study to bracket the thermal failure of each bulkhead classification. In addition, the comparison to the actual data has shown that the analyses in general agreed within 15% of the actual results. This provides a basis for validation of the finite element method, as applied to the thermal analysis of an insulated boundary subjected to fire conditions.

References


Acknowledgement

This work was funded by the United States Coast Guard, and under the direction of Mr. Louis Nash, USCG Research and Development Center, Groton, CT. The advice and comments of Mr. Nash has been instrumental for the success of the current program.
PAPER WITHDRAWN
PAPER WITHDRAWN
GROUND-FAULT CIRCUIT-INTERRUPTER (GFCI)
TECHNICAL REPORT

A. Albert Biss, P.E.

U.S. Consumer Product Safety Commission
Directorate of Engineering Sciences
Division of Electrical Engineering

Over 1,000 deaths and several thousand injuries due to electrical shock and electrical fire hazards involving consumer products occur in the home each year. In addition to these deaths and injuries, property damages due to electrical fires amount to over $1.2 billion each year. The consumer products involved in these incidents are the same kinds of electrical appliances, equipment and wiring systems typical in households across the country.

Considerable effort has been expended in the past by the Consumer Product Safety Commission (CPSC) and others to provide shock hazard protection for consumers in and around the home. First, with each succeeding edition of the National Electrical Code since 1973, protection against shock hazards is extended to various areas of newly constructed homes by requiring the installation of Ground-Fault Circuit-Interrupters (GFCI's). Second, Commission rule-making in the mid-70's required modification of particular citizen band base antennas to reduce consumer exposure to the risk of deadly shock should the antennas contact overhead power lines. Finally, through the voluntary standard system, manufacturers have provided double insulation in various power hand tools.

As a result of these measures, the electrocutions related to consumer products have decreased by 55 percent from 1975 to 1988. This downward trend plateaued somewhat from 1984 to 1986, but more recent data suggest the trend may still be continuing, albeit at a reduced slope. Nonetheless, the death toll is still too high, and further extension of GFCI protection for consumers can effectively reduce electrocutions further, particularly for those consumers residing in older homes. While the Commission's efforts to extend GFCI installation have resulted in shock protection for millions of new homes, the number of older homes left unprotected is still very substantial, amounting to approximately 70 percent of total U.S. households. Thus, a large segment of the population has no GFCI protection and remains at risk from ground fault shock hazards.

Based on CPSC studies, the highest rate of electrical fires occurs in dwellings over 40 years old. The deterioration of aging appliances, equipment and wiring can cause degradation of insulation resulting in current leakage to ground, leading to fire conditions. Because GFCI's detect ground fault currents, they not only protect consumers from electrical shock hazards but they can also prevent some electrical fires from occurring and reduce the severity of others by interrupting the flow of current. Thus, in addition to providing shock hazard protection, GFCI's can significantly reduce the potential for fires, particularly in older homes.

The failure mode leading to ignition in an electrical fire is often difficult to determine since much of the evidence may have been destroyed in the fire. Two mechanisms causing deterioration and breakdown of insulation resulting in ground faults are "arc tracing and treeing" in insulation and "glowing" of loose electrical connections. Electrical tracking involves the development of a surface
conducting path between two points of different electrical potential. This current becomes greater as the thermal degradation of the surface progresses to greater values, leading ultimately to current concentration in a path or channel, sparking, and electric arc-over.

In the opinion of one author, arc tracing or treeing is not a rare process; it is responsible for many electrical fires. One reason for this is that arc tracking usually occurs at low current values, low enough that the overcurrent protection device does not sense it. The process, therefore, continues unhindered until it is detected by the user of the device, until sufficient heat is generated to cause a fire, or until the user discards and replaces the product without realizing that there was a problem with it.

Another mechanism resulting in insulation deterioration is the phenomenon of glowing electrical connections. In its 1977 study of glowing electrical connections, the National Bureau of Standards (NBS) defines "glow" as "a sustained source of light and heat emanating from the vicinity of an electrical connection or from the vicinity of some other place in an electrical circuit, such as where broken parts of a wire, are in contact. The report describes and characterizes the extent to which loose electrical connections in residential type branch circuits have overheated in the laboratory. In electrical devices where connections become loose due to creep, thermal expansion and contraction, corrosion, etc., they may overheat due to their high resistance and/or arcing. The resulting high temperatures can cause insulation to deteriorate thus causing current leakage to ground or enabling the line conductor to contact a grounded conductor or surface. During the Lab experiments at NBS, the GFCI operated twice, thus opening the circuit and disrupting the test. The GFCI was then removed from the circuit, permitting the experiment to proceed.

Because the GFCI detects ground faults, it can prevent some electrical fires and reduce the severity of others by interrupting the flow of electric current. Early detection by GFCI's of ground leakage currents resulting from glowing or arcing due to loose connections and arc-tracking or treeing can prevent major electrical fires from starting.

The activities initiated by the U.S. Consumer Product Safety Commission and others provides a foundation for expanding the participation by local groups, organizations, associations and firms to reach the great majority of consumers not presently protected by GFCIs and who may not be aware of GFCIs and their benefits in the home.
One and Two-sided Burning of Thermally-Thin Materials*

T. Ohlemiller and J. Shields
Building and Fire Research Laboratory
National Institute of Standards and Technology
Gaithersburg MD 20899

Abstract

In spacecraft applications, many slab-like materials are dimensionally thin as a result of weight considerations. In terrestrial applications, thin materials are encountered in clothing, draperies, paper, electronic circuit boards, etc. These materials may be expected to approach thermally-thin behavior during combustion. By thermally-thin is meant that heat absorbed on one surface of the material will penetrate its thickness sufficiently rapidly so that there will no significant temperature gradient through the material depth.

One of the important flammability measures of any material is its rate of heat release per unit area; this is a function of the incident heat flux on the material. Rate of heat release helps determine how fast flames will spread over a material, particularly in a concurrent mode. The total rate of heat release is also one of the best measures of any fire involving the material.

The rate of heat release from a thermally-thin material burning on both sides is shown here to be enhanced by more than the simple factor of two expected for thermally-thick materials. (That is, when both sides of a thin material are burning, there is an enhancement in burning rate beyond that due to the simple doubling of the burning area.) Two simplified models, one based on the artifice of a steady-state burning process and the other accounting for transient fuel consumption, demonstrate that this is a consequence of the Arrhenius temperature dependence of the gasification rate of the solid. In effect, the condensed phase responds to the increased heat input (from the flames on the second side) by raising its temperature and thus its gasification rate. The models, with typical parameter values as inputs, predict a two-sided rate of heat release more than three times that of one-sided burning; see Figure 1. (There is a complication in the experimental demonstration of this effect that lies behind the fact that there are two curves in Fig. 1 labelled "one-sided". The most appropriate comparison with the two-sided case is based on the lower, dotted curve; see below.)

*Sponsored by NASA Lewis Research Center, Order #C-32003-R
In order to demonstrate this effect with real materials, it was necessary to modify the NIST Cone Calorimeter so as to permit two-sided irradiation of thin samples. The Cone heater was replaced with two identical flat radiators with a sufficient area so as to permit essentially uniform heating of samples that were 8.25 cm square. The two radiators were vertical and parallel with a gap of 3.2 cm in the center of which was the sample, atop a weighing cell. The samples could be irradiated on one side and insulated on the back, thus giving the normal type of Cone exposure, or they could be irradiated and then ignited simultaneously on both sides. The materials used were three resin/glass fiber composites, two with polyester resin and one with epoxy resin. All were 0.16 cm thick.

In this experimental arrangement there is no way to prevent top edge burning of the gases coming from the back of the sample during one-sided irradiation. That is, these rear surface gases meet the flames coming up from the front surface along the top edge of the sample holder and burn there. This burning provides no heat feedback to the rear face of the sample. The apparent rate of heat release is thus about of factor of two higher than that for true one-sided burning. In Figure 2 below this effect is contained in the data labelled "one-sided".

The experimentally measured enhancement of rate of heat release is somewhat more than the factor of three predicted by the models. It is inferred that this heat release enhancement means that thermally-thin materials will undergo upward flame spread more readily when burning on two sides than when burning on one side.
A Comparison Of Numerical and Exact Solution of the Creeping Flame Spread Over Thermally Thin Material

Subrata Bhattachajee
Department of Mechanical Engineering
San Diego State University, San Diego, CA 92182-0191

ABSTRACT. A numerical solution of the well known de Ris (1969) problem of flame spread over thin condensed fuel in opposed-flow environment is presented. The problem is reformulated in terms of four non-dimensional parameters, \( \beta_1 \equiv \frac{m_{\text{max}}}{S}, \beta_2 \equiv \frac{\Delta H}{C T_\infty}, \beta_3 \equiv \frac{\dot{T}_v}{T_\infty}, \text{ and } \beta_4 \equiv \frac{\dot{E}_{\text{in}}}{C T_\infty}, \) retaining all the assumptions of the original theory. For a particular baseline case \( (\beta_1 = 0.45, \beta_2 = 26.94, \beta_3 = 2.18, \text{ and } \beta_4 = 0.45) \) which correspond to a 50-50 oxygen/nitrogen environment, cellulosic fuel \((C_6H_{10}O_5)\), and a vaporization temperature of \(650K)\) eleven sets of calculations are performed to check the accuracy of the solution, the grid independence and domain-size independence; the resulting flame shape and the nondimensional heat flux are shown in Fig. 1.

The results of this numerical analysis indicates that the flame is lifted despite the linearization of mass-flux at the vaporizing surface. The non-dimensional flame stand-off distance vary with the controlling parameters in a way similar to the behavior exhibited by one-dimensional diffusion flames. While de Ris theory uses the approximation that the location of the leading edge and the location of the onset of vaporization, the eigen location, are identical, the present analysis treats the leading edge as part of the solution. The most important finding of this work is that the location of the flame leading edge is usually upstream of the eigen location, \( x=0 \), which has significant bearings on the spread rate formula.

The value of \( A \) in the de Ris formula for the spread rate, \( \dot{T}_v \equiv \frac{\dot{E}_{\text{in}}}{C T_\infty} \dot{V}_f = A \frac{\dot{T}_v - \dot{T}_x}{T_\infty - T_x} \), is found, approximately, to be about unity in the present work rather than \( \pi/4 \), suggested by Delichatsios (1986). This is shown in Fig. 2 where the non-dimensional spread rate, \( \dot{\beta} \), is plotted against the de Ris parameter for a wide range of values of the controlling parameters. It can be seen that the value of \( A \), the slope, is sensitive to the vaporization parameter, \( \beta_4 \equiv \frac{m_{\text{max}}}{C T_\infty} \), and to some extent to the stoichiometric parameter, \( \beta_1 \equiv \frac{m_{\text{max}}}{S} \). Parameters that affect the location of the flame leading edge the most. When the values of the controlling parameters are so chosen that the flame leading edge is very close to the eigen location, the value of \( A \) approaches \( \pi/4 \). The results of the present analysis suggests that the exact solution is valid for these limiting values of the controlling parameters.

References:
Fig. 1. Flame shape and the non-dimensional heat flux for eleven sets of calculations varying grid spacing, grid geometry, and domain size.

Fig. 2. Non-dimensional spread rate plotted against the de Ris parameter. Variation of $\beta$ with $\beta_4$ does not appear to be captured well by the de Ris parameter.
The Behavior of High Density Cellulose Samples
Under Simulated Fire Conditions

I. Milosavljevic and E.M. Suuberg
Division of Engineering, Brown University
Providence, RI 02912

INTRODUCTION

The behavior of high-density samples pressed from high-purity cellulose powder has been examined under conditions that simulate those that exist in fires. The objective of this work was to establish what factors are most important in determining the yields of char and volatiles, and to evaluate how well these can be predicted from simple TGA-type of measurements of cellulose pyrolysis. In this paper, the effects of density, sample size, and "grain orientation" are discussed.

EXPERIMENTAL

The pyrolysis behavior of pressed cellulose samples of various densities was examined in three different devices. The first is a standard thermogravimetric (TGA) device. This device has been classically used to establish the kinetics of pyrolysis of cellulose powders, from which the behavior of large samples under fire conditions has been predicted. Here, the technique was modified only insofar as slightly larger samples were used than is customary (3 mm cubes). The reason for using these large particles was a desire to see the effects of sample density in the TGA device. All the samples were pressed from the same powder, and thus the actual chemistry of pyrolysis should be invariant.

The second device employed was a "heated wire mesh" reactor in which thin wafers of pressed powder could be examined. In this device, the sample is heated by placing it within the folds of a stainless steel wire mesh that acts as a resistance element within a circuit. Volatiles escape from the mesh into a cold surrounding gas, and are quenched. This device has been earlier shown to give pyrolysis kinetics that are in excellent agreement with those from standard TGA devices. Here, the interest was in the effects on devolatilization of varying a single sample dimension.

The third experimental device was a simulated fire apparatus, in which large samples (38 mm diameter and about 10 mm thickness) are radiatively heated from the front face. The course of pyrolysis is followed by continuously recording the mass of the sample, as well as its interior temperatures. The samples in this study were exposed to a surface flux of 40 kW/m², typical of fire level fluxes.

The pressed cellulose samples were made in house by pressing high purity filter pulp in a laboratory press. The samples have been recently shown to be good surrogates for the behavior of wood, as they have comparable densities and show comparable behaviors.¹

RESULTS

The experiments in the TGA device have revealed that if heating rates comparable to those in fires are to be used to study the kinetics of pyrolysis, there is a significant danger of external heat transfer

limitations, with particles even a few millimeters in size. Since use of such large particles is not common practice in TGA studies, this is not of particular concern, unless the sample bed size itself reaches such dimensions. Of greater concern was the observation that there is a measurable effect of density on the yields of volatiles, even with particles a few millimeters in size.

In the measurements performed in the heated wire mesh apparatus, it was observed that the escape of volatiles is sensitive to direction of transport. In the pressing of the high density cellulose samples of interest (1 g/cc), a banded structure is developed in the direction perpendicular to the pressing direction. This banded structure is analogous to the grain structure of wood. Volatiles apparently travel for quite large distances along the grain, in preference to crossing it.

Finally, in the simulated fire apparatus, the behavior is strongly influenced by both heat and mass transfer limitations. Heat transfer in the simulated fire situation is controlled by conduction into the bulk of the sample. As a result of the imposed radiative flux condition at the surface, the sample surface continually heats up, and the pyrolysis wave is driven at roughly constant rate into the sample, up to the point at which heat losses from the back balance the input from the front, and the process stops. Higher density samples thus pyrolyze at higher rates, by virtue of their better conductivities. These samples show a lesser temperature gradient across their thickness for the same reason. Mass loss rates are not significantly affected by grain orientation, since the rates are primarily determined by heat transfer rates. The effects of mass transfer limitations are however clearly visible in these large samples. Volatiles produced in the hot forward zones of the sample can diffuse backward, recondense, and later act as heat sinks, when the pyrolysis wave reaches points further back in the sample.

Standard TGA kinetics do not predict well the behavior of the devolatilization in the latter part of the process, in parts of the sample far from the surface. Earlier studies in this laboratory indicated that heating rate has only a minor effect of the global kinetics of pyrolysis of cellulose powder. The present study suggests that the conclusion must be carefully reconsidered, in view of the complications caused by the mass transfer limitations within the sample. The kinetics are, in fact, path dependent for reasons not yet fully understood.

The “char yield” from the simulated fire experiments is not a well-defined quantity, influenced as it is by heat losses from the rear face of the sample. Char yield increases with increasing density of sample, in a direction opposite the trend that would be predicted from the better thermal conductivity of the higher density samples. Thus there is unquestionably an influence of mass transfer on the ultimate yields of char. The effects of density are, again, small, as they were in the experiments from the TGA.

CONCLUSIONS

The results have clearly demonstrated that prediction of the behavior of cellulose pyrolysis under simulated fire conditions is difficult, based upon TGA results alone. There are issues related to correct prediction of heat and mass transfer effects that cannot be addressed from such data alone. A comprehensive model of the behavior of pyrolyzing bulk solids is under development and incorporates many of these issues.

The NIST Annual Conference on Fire Research has long been the prime forum for the presentation and discussion of the latest advances in the science of fire and the engineering of fire safety. This year's conference has been expanded to include all fire research performed within Federal laboratories or sponsored by Federal agencies, as well as by laboratories around the world.

fire research, fire plumes, flame spread, fire suppression, mathematical models, halons, microgravity, fire detection, polymers, vents, sprinklers, pool fires, soot, toxic gases