NHTSA 98 - 3588

NHTSA-98-3588-36

52608

APPLICATIONS OF THE FINITE VOLUME RADIATION MODEL TO PREDICT FIRE BEHAVIOR IN OCCUPIED SPACES

99 MAR 22 AM 10: 34

SERVICES DIV.
RECEIVED

James A. Ierardi (ierardi@wpi.edu) Richard D. Pehrson (pehrson@wpi.edu), and
Professor Jonathan R. Barnett (jbarnett@wpi.edu)
Center for Firesafety Studies, Worcester Polytechnic Institute, Worcester, Massachusetts, USA

INTRODUCTION

The prediction of the growth of a hostile fire in a space such as a room or a transport vehicle is complicated because radiant energy is the driving force behind fire growth and spread for solid fuels. In addition, the radiant energy becomes a direct source of injury to people that are in close proximity to an uncontrolled fire in an occupied space. The study of uncontrolled fires in spaces such as buildings and passenger vehicles was done using the computational fluid dynamics code TASCflow (Advanced Scientific Computing; Waterloo Canada) to predict the effect of radiation on the contents and occupants of such spaces.

In this study, TASCflow was used to make local predictions of temperature and species concentrations for an uncontrolled fire. An important sub-model used was the eddy dissipation concept, which controls the rate of reaction between fuel and oxidizer using a single step reaction. The production rates of soot and fire gases (CO_2, CO, etc.) were based on bench-scale test data for solid fuels for which the combustion reactions can be difficult to predict from first principles.

Other sub-models were used to predict radiation transport properties throughout the non-isothermal fluid domain. For example, local values of absorptivity for a given control volume were evaluated with band models ABSORB and RADCAL. These models are commonly used in fire protection engineering calculations.

Energy transport prediction requires some type of simplification and approximation to overcome the computational expense associated with a full band continuous solution. Optically thin and optically thick models are available, but are insufficient for use in most fire predictions due to the intermediate behavior of fire gases and soot. The discrete transfer method has been widely used in the area of fire predictions to model energy transport. An advantage of the fully conservative form of the finite volume transfer method that was used in this study is that the same grid as the fluid solution can be used. This results in a fully conservative formulation.

Current research to be presented includes use of TASCflow to predict the heat transfer and fire spread from the engine compartment to the passenger compartment for post-crash vehicle fires. In addition, TASCflow was used to predict fire growth on upholstered furniture. This commonly ignited fuel in the residential fire situation has been examined to better understand the effect of an uncontrolled fire on the contents of a building as well as on the life safety of the occupants.

ROLE OF RADIATION IN HOSTILE FIRES

The problem of calculating radiation transfer in a fire can be divided into two parts. First, the local temperatures and soot concentrations must be determined. Second, some type of model must be employed to calculate the emission, absorption and scatter of incident radiation.

Throughout the discretized fluid domain, a statement of conservation of energy is applied at every node location. While solving the energy equation to eventually obtain the temperature distribution, the divergence of the radiant flux appears in the energy equation as a source term.

The products of combustion above a burning fuel are non-homogeneous when evaluated at length scales on the order of the problem geometry. Following from the definition of a control volume, the fluid occupying each control volume is considered homogeneous for the calculation of radiation transfer and properties. To allow radiation properties (absorption coefficient and emissivity) to vary as a function of local species concentration throughout the computational domain, two different property models have been implemented.

The Eddy Dissipation Model

The Eddy Dissipation model assumes that the chemical reaction rate for combustion is small when compared to the rate of the small-scale mixing rate for the reactants [1]. The small-scale turbulent mixing rate is expressed in terms of the turbulent kinetic energy, k, and the dissipation rate, ε. In the case of turbulent diffusion flames, the assumptions of the Eddy Dissipation model are valid because the time scale of the combustion reaction is shorter than the time scale for the oxygen from the surroundings to turbulently mix with the gaseous volatiles.

Determination of Radiation Properties

Analytical tools such as ABSORB and RADCAL utilize both experimental data and theoretical approximations to calculate radiation properties of gases involved in combustion processes. ABSORB is based on a model developed by Modak [2] and estimates the absorptivity and emissivity of homogeneous isothermal mixtures of CO_2, H_2O, and soot using the results of an exponential wide band model. RADCAL, developed by Grosshandler [3], is a narrow band model that calculates radiation characteristics of a gas based on theoretical approximations and tabulated spectral properties. While it offers a greater degree of accuracy than wide band models in predicting radiation properties, it comes at the expense of a significant increase in computational time.

This data can then be used for specifying the combustion reaction of the fuel in conjunction with a radiation model to simulate a fire in an occupied space. The nature of simulating the complex combustion, radiation, and fluid flows of such fires requires the level of sophistication that computational fluid dynamics offers.

MODELING RADIATION

Although several computational fluid dynamics software packages provide sufficient sub-models that allow for the proper modeling of hostile fires, TASCflow [4], produced by Advanced Scientific Computing, has been chosen to model fires in occupied spaces.

Some of the sub-models used in TASCflow, such as conjugate heat transfer (CHT), a polymer combustion model, and a graphical user interface for visualization of results, make it a desirable software package. Additionally, there are several radiation sub-models available that include optically thick, surface to surface, and finite volume radiation methods.

Finite Volume Radiation Model

The radiation from a hostile fire exhibits an intermediate behavior that cannot be accurately described by either an optically thick model or a surface to surface radiation model. However, the finite volume approach has the ability to account for the intermediate behavior of a fire in an occupied space that can be used in conjunction with soot production data to provide a more accurate representation of fire growth. The finite volume radiation transfer model that has been implemented in TASCflow is based on work done by Raithby and Chui [5,6] as well as by Chai *et al.* [7].

This paper will discuss two fire scenarios where an enhanced radiation algorithm is needed. The first is the fire spread in post-crash vehicle fires, the second is fire spread and growth in common household furniture.

FIRE SPREAD FROM ENGINE TO PASSENGER COMPARTMENTS

The objective is to determine the most likely paths of fire spread from the engine compartment into the passenger compartment. With this information, computational fluid dynamics can be utilized to model these situations and provide some insight and a framework for investigating possible firesafety improvement measures.

Post-Crash Vehicle Fires

The engine compartment fire scenario is most likely because the engine compartment contains a variety of combustible materials (gasoline, motor oil, plastics), numerous hot surfaces (manifold), and long lengths of electrical wires that can be compromised to cause a spark when the engine compartment is displaced in a collision. The spread of flame and toxic combustion products from the engine compartment to the passenger compartment involves a risk to the life safety of vehicle occupants. Two barriers exist between the engine and passenger compartment; the bulkhead and the windshield.

Inherent in its nature, the windshield offers less resistance to flame spread and fluid flow of an engine compartment fire than the bulkhead does. Therefore, the behavior of the windshield under these conditions is essential to understanding the spread of fire from engine to passenger compartments in post-crash vehicles.

Phenomena Modeled

The first step towards understanding the spread of flame from engine to passenger compartments in post-crash vehicle fires was modeling the bulkhead [8]. The model included two layers of insulation enclosing a layer of steel subjected to a time varying heat source that is consistent with a standard furnace heat curve used in ISO 834 [8], a standard test for determining fire resistance. The conduction phenomenon in the bulkhead model was compared to the two-dimensional finite element model TASEF-2, which is typically used to model steel structures subjected to furnace fire testing. TASEF-2 only accounts for conduction in a two-dimensional solid and does not model conjugate heat transfer.

The bulkhead model was constructed in a fully parametric fashion, allowing any arbitrary geometry and material. The resulting model can then be compared to experimental data or included in a composite model representing the engine and passenger compartments of a post-collision vehicle.

The second phase of the project was to model the spread of flame form the engine to passenger compartment of a post-collision vehicle via the windshield [9]. This is a common path of flame spread for the engine fire scenario. However, from a modeling perspective, the behavior of the windshield when subjected to the heat flux and fluid flow of an engine fire is more complicated.

The windshield of a post-collision vehicle is often cracked but, held in place by the plastic interlayer. During a post-crash car fire the heating of the windshield causes the polyvinyl butyral inter-layer to melt and pieces of the windshield to fall down opening holes in the windshield. For modeling purposes, it is represented as a porous object that has fluid porosity that changes from 0 porosity to 100% due to heating and then allows gases to flow through the porous windshield materials between the engine and the passenger compartments.

The TASCflow model used as illustrated in Figure 1, utilizes a multi-grid approach to simulate a windshield subjected to an engine fire.

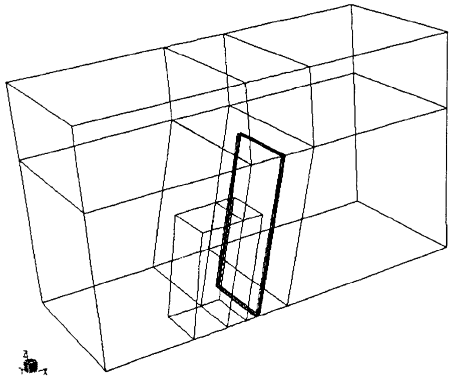

Figure 1--TASCflow model of the windshield.

The windshield object is the bottom center object in the assembly and contains fluid regions attached to the front and back faces using a multi-block grid along with embedding. These two fluid regions are included in order to bias nodes towards the windshield where large temperature and velocity gradients are expected within the fluid domain. Fluid objects are attached to the back, front, and above the windshield object to properly capture the fluid motion in the vicinity of the windshield. Two areas of grid embedding exist in front of the windshield object where the grid is refined for the purpose of properly modeling the combustion that will take place in representing the engine fire.

Boundary Conditions

A symmetry boundary condition has been applied along the entire length of the windshield model in order to reduce the number of nodes required by a factor of two, therefore the fire is assumed to be symmetric about the engine compartment as well. Constant pressure opening ambient boundary conditions have been attached to all external fluid boundary faces. A transient inlet boundary condition is used at the bottom faces of the two embedded regions. A supply of methane (mass specified inlet) is used for a combustion reaction that will approximate the energy release rate associated with a design curve for vehicle fires originating in the engine compartment utilized in designing steel car park structures [10]. Each of the three layers of the windshield is specified as a porous object to approximate the opening of holes in the windshield involved in a post-collision engine fire. The simulations required to model the windshield subjected to a growing engine fire require a transient solution as the temperature, density, velocity values will vary with time.

Initial Results

Initial modeling efforts have begun with a series of simulations to determine the best grid size for the transient windshield problem. The flow field patterns from these simulations have been analyzed for consistency with what would be expected from intuition for an inlet source of fluid encountering a solid object.

Future Work

Once the proper grid size has been determined for a typical engine fire, comparisons can be made to experimental data. Important parameters such as flame and windshield temperatures, the rate of energy release from the fire, and the radiant heat flux levels at specific locations have been collected in experiments.

Acknowledgement: The report was funded by General Motors pursuant to an agreement between General Motors and the U.S Department of Transportation.

FIRE GROWTH ON UPHOLSTERED FURNITURE

Currently a significant effort is being made internationally to improve the ability to predict compartment fire growth on contents and surface linings based on bench-scale heat release data, such as a cone calorimeter (ASTM E1354). The ability to estimate the rate of fire spread using reaction to fire measurements and material properties would provide a quantitative risk assessment tool that can be used for setting fire regulations, design and reconstruction.

The problem of fire growth prediction differs from other forms of combustion modeling (furnaces, engines, power generation etc.) in the prior knowledge about fuel supply. For most combustion situations, the fuel composition and flow rate is known, if not controlled, by the process, simplifying input for the numerical modeler. Contrast this to the situation where it is the heat feedback from the fire itself that controls that rate of fuel generation in an uncontrolled fire, the composition of which is not well defined or known in advance.

A CFD code with an accurate radiation transfer routine and combustion model can provide fluid flow and heat transfer information to a pyrolysis model when the two are coupled. Existing CFD component sub-models for heat transfer, turbulence, combustion and species transport are able to provide boundary condition information at the surface of a fuel. A flame spread routine can use this information to predict pyrolysis, flame spread and ultimately fire growth rate using bench scale and material property data as input. By applying source terms to the conservation equations for mass, momentum, energy and species, a furniture fire model has been developed to interface with the CFD code TASCflow to predict the rate of fire growth on furniture.

Upholstered Furniture Fires

One of the most commonly encountered furniture items in the residential setting, the Polyurethane foam "easy chair", is also the most frequently burned fuel package in the laboratory. Chairs typically have a foam seat and sloped back, along with armrests and are sometimes decorated with buttons, welt cord edging on the cushions and possibly skirting around the base. A number of geometric irregularities such as crevices and gaps are formed which complicate prediction of how the furniture will burn, and due to construction techniques, can vary from piece to piece for the same design. Experimental data shows that such irregularities can have a profound effect on how a chair will burn, especially close to the ignition source and during fire growth.

The ability to gather useful experimental data for modeling purposes has grown significantly over the last 15 years with the refinement of oxygen consumption calorimetry [11]. During a fire, furniture can be expected to burn with local regions of high temperature, velocity and species gradients, a range of time and length scales, and unknown intermediate chemical reactions, all adding more complications to quantification efforts.

The response of furniture and its components to fire can be measured with bench, full and real scale tests to quantify performance according to national and international standards. The goal of these tests is the generation of material fire properties that can be used for modeling, comparison and ranking as characteristic of the particular fuel and not the specific geometry.

By far, the most complete furniture fire test program is the recent Combustion Behaviour of Upholstered Furniture (CBUF) research effort [12] conducted under the European Commission on Measurements and Testing. The goal was to develop standardized quantification methods to measure the fire behavior of furniture to support the harmonization of European regulations. Modeling environmental conditions within the furniture calorimeter necessitates careful selection of boundary conditions, Figure 2.

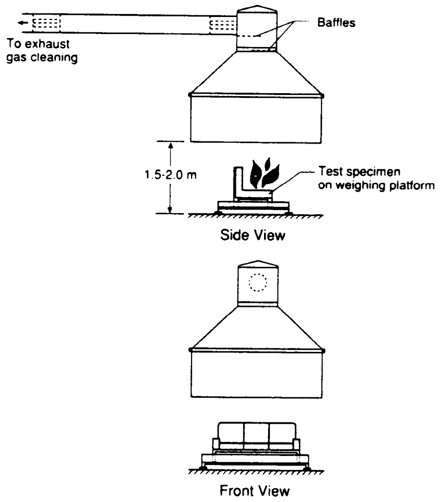

Figure 2. Furniture Calorimeter-Full Scale Test

This real scale test measures the same data as the cone calorimeter, but in this case for a full-scale piece of furniture. The hood above the sample collects the products of combustion which are analyzed to ultimately determine the heat release rate and species concentrations.

Phenomena Modeled

A furniture CFD fire growth routine has been incorporated into TASCflow to generate transient boundary conditions for pyrolysis from a burning surface [13]. At the start of a model run, fuel surfaces are specified in the model, along with ignition criteria, thermal properties, and the bench scale test data for fuel mass loss rate and soot yield as a function of flux. At the start of a run and at each new time step, all solid fuel surface nodes on the boundary interface with the fluid are visited to evaluate ignition status. Nodes that satisfy the ignition criteria (temperature or radiant flux above a user specified critical value) are considered to be pyrolyzing and the time of ignition noted. For all ignited nodes, source terms for fuel, momentum and energy are added to the fluid control volume above the burning node (Figure 3). Burnout is considered when a node has released total mass equal to that measured in the cone calorimeter.

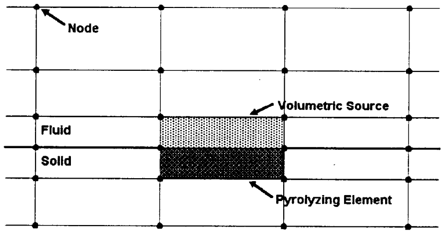

Figure 3. Volumetric Source Term Application

Solution of the conservation equations at fluid nodes throughout the computational domain determines the rate of fuel combustion and resulting fluid motion, soot production and heat transfer. The most important result produced for flame spread is the radiant heat flux to the wall, which provides the driving force for evaluating the rate of fire growth. Coupled with the fluid flow is a simultaneous solution for energy transfer in the solid fuel as part of the process in determining the wall flux. The fuel is treated as an inert solid, a similar simplification employed by many thermal models of flame spread.

The fluid above the solid fuel provides a region where the combustion reaction can take place. Once the pyrolysis products are released from the surface, they are in a position to mix with air while being heated from the nearby flame. Initially the fuel occupies a region that is too rich to burn, but the action of mixing and diffusion reduce the concentration of fuel to a point where it can burn.

A complex mixture of pyrolysis products are produced by the heating of foam and fabric during a furniture fire, with the actual rate of consumption of fuel depending on both the reaction rate and the speed at which fuel and oxidizer are mixed. Once heated to ignition, oxidation of the fuel leads to the generation of products of combustion. The intermediates and products of combustion are an equally complex function of the rate of reaction and local conditions.

Boundary Conditions

The spread of flame across a surface is a transient process represented by a continuous advancement of the flame front(s). Upon reaching a sufficient temperature, the solid will begin to lose mass through pyrolysis. The greater the heating through the action of external flux or solid conduction, the larger the value of mass loss. Production rates of all species are simply based on those measured in the cone calorimeter. This allows generation of CO, H_2O, and soot to be a transient function of external flux in the form of yield rates.

Modeling the ignition process is a complex task requiring a coupled solution for unsteady physical and chemical processes in both the gas and solid phases. Two criteria will be used to predict ignition of a fluid-solid boundary face. Based on the solid heat conduction calculations, a surface temperature on the solid at the fluid interface is calculated. When this temperature exceeds the ignition temperature for the fabric, ignition is assumed. The radiation heat transfer model calculates the flux incident upon each flux element face, a second option for specification of ignition. When the external flux level reaching the surface exceeds the critical flux, ignition results.

Below the hood of the furniture calorimeter, flow conditions into the calorimeter around the perimeter are not known in advance, making it difficult to specify a pressure distribution across this plane. Further from the calorimeter, the air in the laboratory should be reasonable quiescent, with ambient properties of pressure and temperature. Boundary condition specification is made by extending out the computational domain away from the hood to a point where an ambient pressure opening boundary condition can be applied. The downside to this scenario is that the boundary condition statement becomes very weak, especially in the far corners removed from the forced flow induced by the extraction fan. The robustness of the CFD code and solver are tested fully since convergence will suffer due to the formation of regions (pockets) of poorly reduced residuals which can stall out a transient run. The flow rate of combustion products through the top of the hood induced by the extraction fan is also unknown in advance, although it is measured during testing.

Results

The fire growth routine coupled to TASCflow is currently being tested against full-scale upholstered furniture and mattresses data measured during the CBUF program. Preliminary results from the fire growth model coupled with the finite volume radiation model show that the ignition process resulting from a gas ring burner can be properly modeled. Radiant flux levels measured from the burner agree very well with those predicted. The rate of flame spread across solid foam mattresses, and resulting heat release rate also agree well with experiment.

CONCLUSIONS

Radiation plays an important role in the growth and spread of fire involving solid fuels in occupied spaces. The ability to accurately model radiation using CFD depends upon the appropriate soot properties and a radiation sub-model that accounts for the intermediate behavior of combustion products in turbulent diffusion flames. The application of the finite volume radiation model in conjunction with the proper determination of soot properties has the potential for improving predictions of fire growth in occupied spaces as demonstrated in preliminary results of research in the areas of post-crash vehicles and upholstered furniture. These efforts will continue with an emphasis on validating the improvement in fire growth and spread predictions made with the use of the finite volume radiation model.

REFERENCES

[1] Cox, G. *Combustion Fundamentals of Fire,* Harcourt Brace & Co., London, 1995.

[2] Modak, A.T. "Radiation from Products of Combustion", *Fire Research*, Vol 1, pp 336-361, 1979.

[3] Grosshandler, W.L. "A Narrow Band Model for Radiation Calculations in a Combustion Environment", *NIST Technical Note 1402,* Building and Fire Research Laboratory, NIST 1993.

{4] *TASCflow3d User Documentation*, Advanced Scientific Computing Ltd., Waterloo, Ontario, Canada, 1992.

[5] Raithby, G.D., and E.H. Chui. "A Finite-Volume Method for Predicting Radiant Heat Transfer in Enclosures with Participating Media", *Journal of Heat Transfer*, Vol. 112, pp. 415-423, 1990.

[6] Chui, E.H., and G.D. Raithby. "Implicit Solution Scheme to Improve Convergence Rate in Radiative Transfer Problems", *Numerical Heat Transfer, Part B*, Vol. 22, pp. 251-272, 1992.

[7] Chai, J.C., H.S. Lee, and S.V. Patankar. "Treatment of Irregular Geometries Using a Cartersian Coordinates Finite-Volume Radiation Heat Transfer Procedure", Vol. 26, pp. 225-235, 1994.

[8] Wittasek, N.B., Pehrson, R.D., and J.R. Barnett, "Computational Fluid Dynamics Modeling of Post-Collision Vehicle Fires", Worcester Polytechnic Institute, Worcester, MA, USA, 1997.

[9] Ierardi, J.A. "Computer Model of Fire Spread from Engine to Passenger Compartments in Post-Crash Vehicle Fires", *M.S. Thesis (work in progress),* Worcester Polytechnic Institute, Worcester, MA, USA, 1998.

[10] Cajot, L.G., *et al.* "Development of Design Rules for Steel Structures Subjected to Natural Fires in Closed Car Parks", ProfilARBED Research Center, Luxembourg, 1996.

[11] Babrauskas, V. and Grayson, S. J. *Heat Release in Fires,* Elsevier Applied Science, 1992.

[12] Sundstrom, B. "Fire Safety of Upholstered Furniture - the final report on the CBUF research programme", European Commission Measurements and Testing, 3478/1/0/196/92/11-BCR-DK(30), 1995.

[13] Pehrson, R.D. "A Pyrolysis Model for Upholstered Furninture", *Ph.D. Dissertation (work in progress),* Worcester Polytechnic Institute, Worcester, MA, USA, 1998.