

Fire Suppression Modeling & Simulation Framework for Ground Vehicles

Vamshi Korivi and Steven McCormick
 US Army TARDEC

Steven Hodges
 Alion Science & Technology

ABSTRACT

The US Army Tank Automotive Research, Development and Engineering Center (TARDEC) has developed a unique physics based modeling & simulation (M&S) capability using Computational Fluid Dynamics (CFD) techniques to optimize automatic fire extinguishing system (AFES) designs and complement vehicle testing for both occupied and unoccupied spaces of military ground vehicles.

The modeling techniques developed are based on reduced global kinetics for computational efficiency and are applicable to fire suppressants that are being used in Army vehicles namely, bromotrifluoromethane (Halon 1301), heptafluoropropane (HFC-227ea, trade name FM200), sodium-bicarbonate (SBC) powder, water + potassium acetate mixture, and pentafluoroethane (HFC-125, trade name, FE-25). These CFD simulations are performed using High Performance Computers (HPC) that enable the Army to assess AFES designs in a virtual world at far less cost than physical-fire tests.

This methodology is applied to vehicle crew compartments for multiple scenarios using HFC227e + SBC powder which is the suppressant combination used in most US combat and tactical vehicles with crew fire protection systems. Predicted and test results match qualitatively very well for overall suppression time as well as for soldier survivability from thermal injury, blast overpressure and inhalation toxicity risks.

After gaining confidence with crew compartment simulations, this fire suppression modeling methodology is now being applied to the geometrically-more-complex military vehicle engine compartments with HFC-125 fire suppressant that is widely used for unoccupied spaces. Challenges associated with the fire suppression simulations are discussed along with future developments that are being proposed to enhance the overall accuracy of the simulation methodology.

CITATION: Korivi, V., McCormick, S., and Hodges, S., "Fire Suppression Modeling & Simulation Framework for Ground Vehicles," *SAE Int. J. Trans. Safety* 5(1):2017, doi:10.4271/2017-01-1351.

1. INTRODUCTION

According to recent data available [1], approximately 1.5% of attacks on US Army vehicles in Iraq and Afghanistan resulted in fires. A total of 220 casualties were attributed to combat fire in the period from 2002 through 2012. These casualties, comprised of wounded in action (WIA) and killed in action (KIA), are summarized in Table 1. Additionally, dozens of accidental/peacetime vehicle fires occur annually and some result in casualties: 33 injuries and 7 fatalities were a result of 35 accidental/peacetime fires that occurred in the 2002 through 2012 period. To reduce these fatalities, military ground vehicles are equipped with automatic fire extinguishing systems (AFES) that protect the crew compartments, engine compartments, and in some applications, tires and/or fuel tanks. Fires in ground combat vehicles may involve petroleum, oil, and lubricant (POL) fuels and/or electrical components or ammunition.

Table 1. Summary of US combat casualties attributed to fire.

Theater	Attacks	Fires	WIA	KIA
OEF*	17,970	240	62	25
OIF/OND**	25,828	401	56	77
Total	43,798	641	118	102

* Operation Enduring Freedom, 01 Jan 2007 - 31 Dec 2012

** Operation Iraqi Freedom/Operation New Dawn, 01 Jan 2002 - 31 Dec 2012

The M&S methodology discussed in this document is restricted to POL-based fires. Having access to M&S results can help designers evaluate tradeoffs relatively inexpensively and quickly. M&S can complement testing by providing insight that leads to fewer physical tests needed to optimize the design of fires inside the crew compartment.

compartment are typically due to an overmatching threat (a combat event that penetrates the vehicle) that involves fuel or hydraulic fluid resulting in a fireball [2]. Engine compartment fires are mainly due to a threat penetrating a fuel cell or hydraulic reservoir (combat fires), or due to a ruptured hydraulic line spraying POL liquid on to hot surface such as a turbocharger, and/or POL fluid collecting in the engine bilge area and ignited due to another fire or machinery failure (i.e., peacetime fires).

A step by step procedure is presented in this paper to design and assess the effectiveness of the AFES using digital simulations. The procedure that is outlined includes calculating the amount of suppressant required for the volume of interest; analyzing the impact of AFES plumbing and nozzles on suppressant discharge, including discharge split among nozzles if there is more than one; determining nozzle discharge characteristics and specification as a boundary condition for 3-D simulation; virtual assessment of fire suppression in 3-D with chemistry for overall assessment of the system to meet the requirements.

In order to limit crew incapacitation, crew compartment fires need to be suppressed very quickly. Crew casualty criteria [2] for occupied spaces are critical and address blast overpressure, thermal injuries, toxic gases, agent concentration, oxygen levels, and fragmentation. To assess the crew casualty using the criteria in the digital simulations, a generalized, reduced chemical kinetics framework was developed that is applicable to any of the suppressants that are being used by the US Army in fire protection systems deployed on ground vehicles [3].

The framework is implemented using CFD techniques and was validated against measurements made using an exploratory test box, and then applied to a typical tactical vehicle compartment. Simulations are done for multiple scenarios including various nozzle locations, number of nozzles, air flow on & off, hatches open & closed, and different stowage configurations inside the compartment. Simulation results match very well with test results in predicting trends and to some extent, agree quantitatively [4]. The M&S framework developed for fire suppression [3] is general in nature and thus may also be applied to an engine compartment fire suppression event. M&S simulation results for the engine compartments with different scenarios of interest are simulated with HFC-125 suppressant.

A collision involving typical automotive vehicles, such as a car or a truck, often results in significant deformation. Thus, to be effective, a system designed to protect against post-collision fires must include consideration of changes in vehicle geometry. However, in contrast, the structural geometry of armored military vehicles generally do not suffer large deformations due to overmatching event. Typically, a threat penetration results in localized deformation that is small with respect to the vehicle. Thus, optimization of AFES systems for military vehicles does not generally require taking into account the geometry changes before and after the threat penetration event [5 & 6].

This paper is organized as follows: Details regarding the calculation of the amount of suppressant required and other factors that need to be taken into account are given in [Section 2](#). Details regarding the quasi-one dimensional code that models suppressant flow from the extinguisher to the nozzles are given in [Section 3](#). Spray parameter characterization for nozzle discharge are given in [Section 4](#). Suppressant concentration uniformity evaluation is outlined in [Section 5](#). Physics based modeling framework for virtual fire evaluation and application to crew and engine compartment are given in [Section 6](#). Limitations of the methodology are summarized in [Section 7](#). Summary and conclusions are given in [Section 8](#). A reduced set of fire suppression reactions for HFC-227ea & SBC powder and HFC-125 are presented in [Appendix A](#).

2. SUPPRESSANT AMOUNT CALCULATION

Once the suppressant is chosen for a specific application, determination needs to be made for the amount of agent that is needed for effective suppression. [Equation \(1\)](#) helps determine the amount of agent required at a given temperature.

$$C = \frac{100}{\frac{V}{WS} + 1} \quad (1)$$

C = agent concentration (%)

V = compartment volume (m³)

W = agent weight (kg)

S = specific vapor volume at T (m³/kg)

The minimum amount of agent required is based on the application and agent type as defined in applicable NFPA documents. One also needs to take into account the change of concentration due to environmental factors such as temperature, elevation and stowage [7] since these factors can significantly impact the amount of agent required.

3. SUPPRESSANT DISCHARGE FROM QUASI 1-D MODEL

Once the suppressant amount is determined based on the protected volume and environmental conditions, one has to determine the agent discharge time, and, if there are multiple nozzle, the split among nozzles, and the discharge lag among the nozzles taking into account plumbing effects. A typical single-phase flow network model cannot be used for this application as the suppressant is stored at high pressure with dissolved nitrogen and the suppressant discharge involves a dynamic two-phase mixture.

The US Army flow model [8] is developed using a quasi-steady approximation to calculate suppressant flow rate for a pressure interval. Thermodynamic properties used in the software are based on Jet Propulsion Laboratory test results [9]. This software can be used to predict the lag time from valve actuation to suppressant discharge taking into account plumbing effects, diameter changes, and

restrictions due to bends, pipe fittings, and nozzle discharge coefficients. Nitrogen solubility is modeled using the Peng-Robinson equation of state with geometric mixing constants for mixture properties. This software predicts, 1) flow splits from multiple nozzles, 2) lag between the nozzles, 3) suppressant quality (vapor mass/ [vapor mass + liquid mass]), 4) suppressant temperature, and 5) suppressant and nitrogen mass flow rates out of the nozzle. The software is validated with experimental data for agents of interest to the Army: Halon 1301, HFC-227ea, water + potassium acetate, and SBC powder.

Typical output from the quasi 1-D software for a HFC-227ea discharge is shown in the Figure 1. Output is plotted for the total amount of agent discharge over 200 milliseconds duration along with the nitrogen content as a function of time. This information is used as boundary condition for CFD fire suppression simulations.

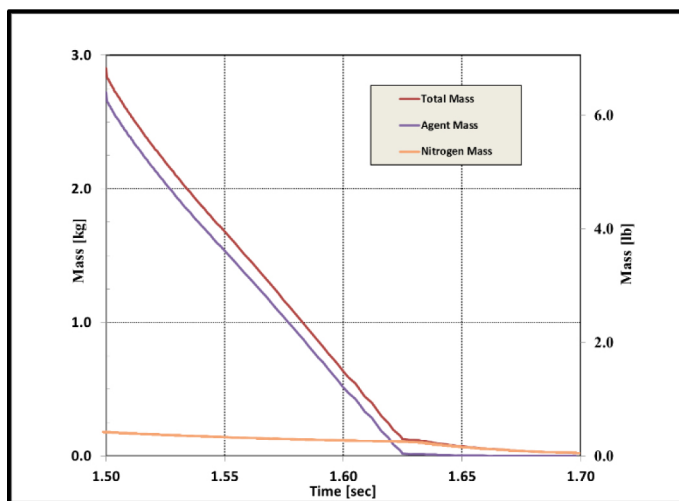


Figure 1. HFC-227ea Mass Flow Discharge Prediction using Army flow model software

4. SUPPRESSANT DISCHARGE PATTERN FROM A NOZZLE

Suppressant discharge nozzle selection is critical in this application. Discharge from the nozzle depends on different factors such as type of suppressant, nozzle design, temperature and pressure inside the bottle. Suppressant discharge characteristics such as spray cone angle, and droplet size distribution, and velocity distribution are vital to how the suppressant is distributed into the volume of interest and eventual interaction with the fireball. The suppressant discharge from the nozzle cannot be determined using a conventional fluid dynamics approach since the length scales involved are much smaller and also, it is challenging to model the dissolved nitrogen coming out of solution from the suppressant, a process which significantly contributes to atomization of the suppressant.

A computational attempt has been made to resolve the suppressant droplet size, velocity and cone angle for a nozzle releasing HFC-227ea pressurized with nitrogen to 62 bar (900 psi). This computational effort was performed to predict the primary breakup spray behavior from the nozzle with a multiphase, Volume of Fluid (VOF) simulation. VOF information was used to process and identify statistical information from the CFD simulation on droplets that are formed during primary breakup. This statistical information can be used to characterize a Lagrangian spray representation of the suppressant from the nozzle.

The VOF computational approach was used to yield the droplet size distribution, velocities, discharge pattern and cone angle. Current state-of-the-art CFD approaches cannot efficiently predict the droplet size and droplet velocity, but discharge cone angle and discharge pattern can be predicted as shown in Figure 2 & 3. The simulated cone angle matched very well with test results as shown in figure 4 for this case. The discharge pattern from simulation seems to indicate that there is a central jet surrounded by a conical spray as shown in Figure 2. Flow split between the central jet and the surrounding conical spray is also determined in the simulation, parameters that cannot be determined from testing.

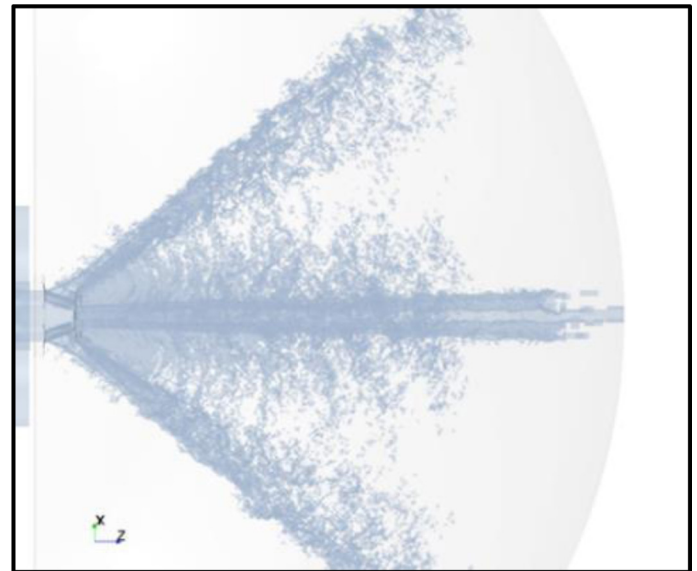


Figure 2. Discharge Pattern from VOF Simulation

The droplet sizes determined in these simulations are around 1000 microns, but strongly depend on the smallest computational cell size that is practical (which is often limited by computational expense).

Our current capability does not allow us to resolve the droplet sizes which have been experimentally observed to be on the order of 50-100 microns. As one cannot resolve the length scales involved in suppressant discharge pattern efficiently, we rely on suppressant discharge testing to measure the suppressant droplet size and velocity distribution. The droplet size and velocity can be determined using experimental techniques such as Phase Doppler Particle Analysis (PDPA) or Particle Image Velocimetry [10].