

REPORT DOCUMENTATION PAGE				Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing this collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.					
1. REPORT DATE (DD-MM-YYYY) 29-11-2006		2. REPORT TYPE Technical paper		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE Progress in Modeling Missile Fuel Venting and Plume Contrail Formation (Postprint)				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) J.D. Chenoweth, K.W. Brinckman, J.J. York, G. Feldman, & S.M. Dash (CRAFT Tech)				5d. PROJECT NUMBER	
				5e. TASK NUMBER MDA0074P	
				5f. WORK UNIT NUMBER	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Air Force Research Laboratory (AFMC) AFRL/PRSA 10 E. Saturn Blvd. Edwards AFB CA 93524-7680				8. PERFORMING ORGANIZATION REPORT NUMBER AFRL-PR-ED-TP-2006-454	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) Air Force Research Laboratory (AFMC) AFRL/PRS 5 Pollux Drive Edwards AFB CA 93524-7048				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S NUMBER(S) AFRL-PR-ED-TP-2006-454	
12. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release; distribution unlimited (AFRL-ERS-PAS-2006-288)					
13. SUPPLEMENTARY NOTES AIAA-2007-1012 ©2007 by the authors. Published by AIAA with permission. Presented at the 45 th AIAA Aerospace and Sciences Meeting and Exhibit, Reno, NV, 8-11 Jan 2007.					
14. ABSTRACT In this paper, we discuss progress made in extending specialized missile plume codes to analyze more generalized problems entailing varied missile propulsive flowfield phenomena. Problems of interest include those of fuel venting and plume contrail formation. To analyze such processes, gas/liquid modeling is being incorporated that includes primary and secondary breakup, and vaporization/condensation physics. This is being performed at an engineering level and overall progress in extending plume codes to analyze these processes will be described. Exemplary problems described include those of both gaseous and liquid fuel venting, application of unified secondary breakup and vaporization of a liquid fuel venting problem, and, contrail formation in a generic missile plume.					
15. SUBJECT TERMS					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT SAR	18. NUMBER OF PAGES 14	19a. NAME OF RESPONSIBLE PERSON Mr. Thomas Smith
a. REPORT Unclassified	b. ABSTRACT Unclassified	c. THIS PAGE Unclassified			19b. TELEPHONE NUMBER (include area code) N/A

Progress in Modeling Missile Fuel Venting and Plume Contrail Formation

J.D. Chenoweth^{*}, K.W. Brinckman[†], B.J. York[‡], G. Feldman[§], and S.M. Dash^{**}
Combustion Research & Flow Technology, Inc. (CRAFT Tech)
Huntsville, AL, Pipersville, PA and Charlotte, NC
Email: jdc@craft-tech.com

ABSTRACT

In this paper, we discuss progress made in extending specialized missile plume codes to analyze more generalized problems entailing varied missile propulsive flowfield phenomena. Problems of interest include those of fuel venting and plume contrail formation. To analyze such processes, gas/liquid modeling is being incorporated that includes primary and secondary breakup, and vaporization/condensation physics. This is being performed at an engineering level and overall progress in extending plume codes to analyze these processes will be described. Exemplary problems described include those of both gaseous and liquid fuel venting, application of unified secondary breakup and vaporization of a liquid fuel venting problem, and, contrail formation in a generic missile plume.

I. Introduction

There are a number of plume/propulsive related problems of great interest to the missile community that are not presently analyzable with engineering-oriented plume codes. These problems include the modeling of events such as fuel venting (occurring during staging or in other scenarios), and, the formation of contrails produced by water condensation (secondary smoke formation) in the plume. The modeling of such events requires the inclusion of gas/liquid methodology into the plume codes, performed at an engineering level to keep the codes fast running and easy to use. The gas/liquid methodology being incorporated is summarized in Table I. This paper will summarize progress made in extending plume codes to analyze varied propulsive related events, focusing on aspects of the gas/liquid methodology being incorporated.

II. Missile Plume Code Features

The principal features of new, engineering-oriented missile plume codes developed by CRAFT Tech are summarized in Table II, with many of the plume specific features having evolved from recent Navier-Stokes code development focused on inclusion of advanced particle methodology¹, turbulence modeling², and turbulent combustion³. These codes have been configured to provide rapid solutions of the complete missile/plume flowfield in a user-friendly manner employing a GUI driven menu. The zonal methodology used is shown in Figure 1 and described in Table III.

The continuum plume codes use a combination of PNS and RANS numerics, implementing RANS methodology with adaptive grids to analyze the base region of plumes and plume-induced separation. They contain turbulent combustion methodology to analyze plumes under conditions where afterburning is marginal. CRAFT Tech has three on-going experimental programs to obtain high-fidelity base-region data, plume-induced separation data, and data in the marginal afterburning regime. These programs are supported by the Army, NASA and MDA and are being performed in collaboration with Calspan University of Buffalo (LENS shock tunnel data – Holden, et al.) and

^{*} Research Scientist, Huntsville, AL, Member AIAA

[†] Research Scientist, Pipersville, PA, Member AIAA

[‡] Principal Research Scientist, Pipersville, PA Member AIAA

[§] Research Scientist, Charlotte, NC, Member AIAA

^{**} President & Chief Scientist, Pipersville, PA Associate Fellow AIAA.

the University of Mississippi (new 12" x 12" trisonic tunnel at the NCPA – Seiner, et al.). A major feature of these plume codes is their ability to perform higher-altitude plume calculations in conjunction with DSMC codes, via calculating a continuum breakdown surface and transferring inflow data from the plume codes to the DSMC code on this surface. This capability has required thermal nonequilibrium extensions to the continuum plume code and compatibility of the thermochemistry and particulate methodology in the continuum and DSMC codes.

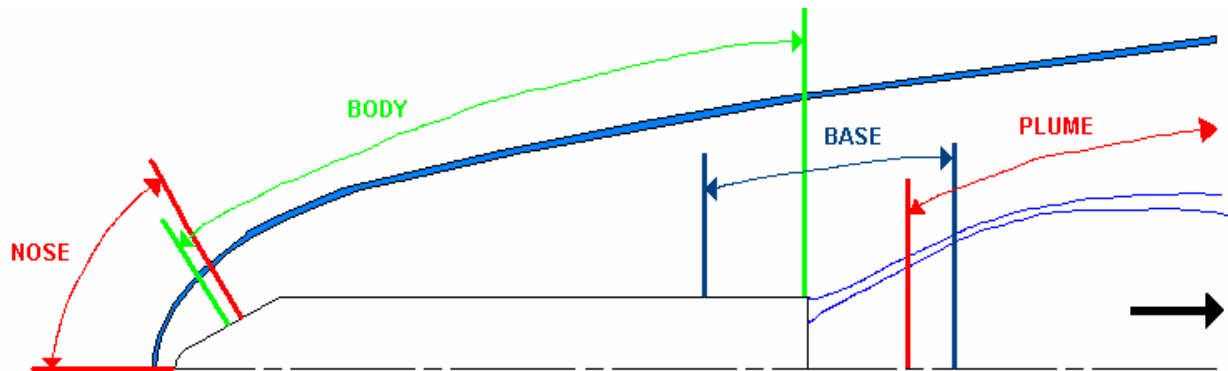


Figure 1. Identification of Major Flow Field Regions for Axisymmetric Missile Flows.

Table I. Gas/Liquid Methodology for Propulsive Event Phenomena

Feature	Relevance	Status
Bulk Liquid/Gas Interactions	Capture interface of vented fuel and its motion.	New pressure-based, preconditioned formulation now operational in CRUNCH CFD [®] , that can analyze real-fluid thermodynamic relations for gas/liquid mixtures. Exploratory studies in progress.
Primary Breakup	Fuel droplets are formed at the gas/liquid interface.	Correlations have been used to provide rate of droplet formulation and droplet sizes but this requires well resolved interface and local properties (surface tension and shear forces).
Secondary Breakup	Larger droplets breakup into smaller droplets.	Droplet Weber numbers which are used to predict where secondary breakup occurs have been determined for simple flows. Breakup methodology now being incorporated into CFD codes.
Droplet Vaporization	Droplets can vaporize, based on heating rates and local conditions.	Nonequilibrium vaporization model incorporated into CFD codes and fundamental studies were performed.
Droplet Combustion	Droplets can burn in hot zones.	Modeling of gas-phase kinetics of fuel vapor available for select fuels.
Condensation/Ice Formation	Water vapor in expanding plume can condense to form droplets, which can freeze.	Basic condensation methodology operational at research level.

Table II. Features of the New CRAFT Tech Developed Rocket Plume Flowfield Models

<i>NUMERICS/ PARALLEL PROCESSING</i>	• GUI-Driven Framework • AXI/3D Finite-Volume Discretization • ADI and L/U, Upwind (Roe/TVD) Numerics • Fully Implicit Source Terms/Boundary Conditions • PNS Spatial Marching Capability • Domain-Decomposition Parallel Architecture with MPI • Preconditioning Extensions
<i>GRID FEATURES</i>	• Self-Contained Grid Generation • Grid Dynamics to Account for Moving Boundaries • Self-Contained Solution-Adaptive Gridding • Noncontiguous Grid Interfacing with Flux Preservation • Zonal NS/PNS Approach
<i>THERMOCHEMISTRY</i>	• Multi-Component Real Gas Mixtures • Finite-Rate Chemistry and Plume Afterburning Database • Fully Implicit Source Term Linearization
<i>MULTIPHASE FLOW</i>	• Nonequilibrium Particle/Droplet Solvers • Breakup Models • Burning, Evaporation, Condensation Models
<i>TURBULENCE</i>	• Specialized Plume Turbulence Model • Scalar Fluctuation Model • Turbulent Combustion Model
<i>RAREFIED FLOW</i>	• Slip Boundary Conditions • Thermal Nonequilibrium • Breakdown Surface Construction for DSMC Interfacing

Table III. Sequential Operation in Axisymmetric Version of the Plume Code

Region	Simulation Methodology
Nose	Nose simulation including adaptation processes run as full elliptic Navier-Stokes
Body	Body simulation either shock fitting or shock capturing run as space marching Navier-Stokes (PNS) with turbulent transition capabilities
Base	Base simulation run as full elliptic Navier-Stokes and including particulates (for SP motors), turbulent chemistry modeling, and grid adaptation
Plume	Plume simulation run as space marching Navier-Stokes (PNS) and including particulates, turbulent chemistry modeling, and high-altitude non-equilibrium effects where needed

III. Gas/Liquid Extensions

A. Overview

The new plume codes are engineering-oriented variants of the CRAFT CFD[®] code ⁴, specialized for missile plumes. Earlier versions of the CRAFT CFD[®] code were developed to analyze liquid propellant guns and bulk liquid interactions with high speed airstreams which utilized Volume-of-Fluid (VOF) methodology ^{5,6}. With VOF methodology, co-volume terms are added to the gas-phase and liquid equations (representing the volume of each computational cell occupied by the gas and bulk-liquid phases respectively) and the thermodynamics of each phase are treated distinctly. Applications of this version of the CRAFT CFD[®] code to varied gas/bulk liquid interaction problems indicated that it operated quite well at elevated pressure levels (as exist in propulsive devices, etc.), but not at lower pressures which are of primary interest for fuel venting problems.

Based on recent work performed for analyzing cavitating flows, a new gas/liquid formulation was developed ⁷ that implements a unified thermodynamic formulation (not requiring the use of VOF methodology) in conjunction

with preconditioning, permitting the code to operate over a broad range of conditions. This newer gas /liquid formulation is operational in the unstructured code, CRUNCH, and its assessment for application to higher speed bulk dispense and venting problems has been performed as part of an IR&D effort described in another paper at this meeting⁸. This same formulation has just been incorporated into our new plume codes with very preliminary results for a liquid jet described in Sections IV.B and C.

Primary breakup providing for the formation of droplets at the “captured” gas/liquid interface was a key ingredient in an earlier version of the CRAFT CFD[®] code and implemented conventional correlations for droplet formation rate and size (see Ref. 6,8 for details). This same methodology is being incorporated into the new plume codes and will be described in a future paper for missile fuel venting applications.

This paper will describe: the basic Eulerian particle/droplet numerics utilized (III.B); the approach used to treat size change due to vaporization/condensation, secondary breakup, and/or combustion (III.C); vaporization/condensation modeling (III.D); and the secondary breakup methodology for droplets (III.E) that is generally used in conjunction with vaporization modeling, as will be described.

B. Eulerian Particle/Droplet Numerics

The standard Eulerian continuum-cloud particulate equations for mass, momentum and energy conservation used to track droplet transport are

$$\frac{\partial Q_p}{\partial t} + \frac{\partial E_p}{\partial \xi} + \frac{\partial F_p}{\partial \eta} + \frac{\partial G_p}{\partial \zeta} = D_p + H_p \quad (1)$$

where:

$$Q_p = \begin{bmatrix} \rho_p \\ \rho_p u_p \\ \rho_p v_p \\ \rho_p w_p \\ e_p \end{bmatrix}; \quad E_p = \begin{bmatrix} \rho_p U_p \\ \rho_p U_p u_p \\ \rho_p U_p v_p \\ \rho_p U_p w_p \\ e_p U_p \end{bmatrix}; \quad F_p = \begin{bmatrix} \rho_p V_p \\ \rho_p V_p u_p \\ \rho_p V_p v_p \\ \rho_p V_p w_p \\ e_p V_p \end{bmatrix}; \quad G_p = \begin{bmatrix} \rho_p W_p \\ \rho_p W_p u_p \\ \rho_p W_p v_p \\ \rho_p W_p w_p \\ e_p W_p \end{bmatrix} \quad (2)$$

The source vector H_p includes interphase drag and heat transfer terms and is given by

$$H_p = \frac{1}{V_{cell}} \begin{bmatrix} 0 \\ A_p (u_g - u_p) \\ A_p (v_g - v_p) \\ A_p (w_g - w_p) \\ B_p (T_g - T_p) \end{bmatrix} \quad (3)$$

The vector D_p contains viscous particle diffusion terms which model the turbulent diffusion of particulates in a manner analogous to the turbulent diffusion of the gas phase. In the current approach, the particle diffusivity is related to the gas phase diffusivity by a time scale relation which locally models the response of the various particulate size groups to the local turbulence field. In the above equations, the gas/particle interaction source terms are functions of the drag coefficient, (C_d), and the Nusselt number (Nu). The current formulation utilizes correlations derived from rocket propulsion applications.

C. Size Change Methodology

In order to analyze the size change of droplets (or particles), an additional conservation equation for particulate surface area is solved, in addition to the total mass of the particle cloud.

$$\frac{\partial S}{\partial t} + \frac{\partial U_p S}{\partial \xi} + \frac{\partial V_p S}{\partial \eta} + \frac{\partial W_p S}{\partial \zeta} = \frac{2}{\rho_p r} \left(\frac{\partial m_p}{\partial t} \right) \quad (4)$$

For an inert particle cloud, i.e. one in which all particles remain the same size, this equation is unnecessary. For particulate clouds that are condensing/vaporizing and/or combusting, where the sizes are not constant and vary in space and time, inclusion of this surface area equation is needed. Particulate cloud surface area is chosen as the additional conserved variable. This along with volume allows us to calculate the Sauter Mean Diameter of the particle cloud, i.e. the diameter of a droplet whose volume to surface area ratio is the same as that for the entire cloud.

$$D32 = 6 V/A \quad (5)$$

Experiments have shown that the Sauter Mean Diameter is an appropriate particle size metric upon which to base vaporization/condensation and/or combustion rate calculations. Application of a Sauter Mean Diameter (SMD) approach to the vaporization/ condensation problem is computationally efficient and avoids the “binning” of particles as the droplets change size. The binning approach is non-continuous, and in the case where droplet radii can range over several orders of magnitude, the binning strategy would require an impractically high number of droplet classes and thus is not practical for engineering calculations.

D. Droplet Vaporization/Condensation Modeling

Methodology has been incorporated into the plume codes that can model vaporization and condensation of liquid droplets. Interphase source terms have been incorporated to transfer mass between phases according to a physically based vaporization/condensation model. Evaporation and condensation of droplets are modeled according to a modified form of the Hertz-Knudsen equation, which gives a mass transfer rate as the difference between incoming (condensing) fluxes from the gas phase, and evaporative fluxes from the droplet.

$$\frac{dm_d}{dt} = N \ 4\pi r^2 \left[\frac{P_{\text{partial}}}{\sqrt{2\pi RT_g}} - \frac{P_{\text{sat_eff}}}{\sqrt{2\pi RT_d}} \right] \quad (6)$$

P_{partial} is the partial pressure of the droplet substance vapor in the gas phase. The modification here is that an effective saturation pressure, $P_{\text{sat_eff}}$, given by the Kelvin-Helmholtz equation, which accounts for surface tension, σ , is used.

$$P_{\text{sat_eff}} = P_{\text{sat}} \exp \left[\frac{2\sigma}{\rho_d RT_d r} \right] \quad (7)$$

Here P_{sat} is the flat film saturation pressure at the droplet temperature and ρ_d is the droplet material density. The droplet material surface tension σ is also expressed as a function of droplet temperature as discussed in the next section. Source terms are constructed to transfer mass between the gas and droplet phases.

The mass transfer is incorporated into the source terms for the Eulerian particle equations to capture the effect of droplet vaporization on the particle cloud response. An additional particle equation tracks the cloud surface area, allowing the model to incorporate variation in the droplet size, which is reflected in the Sauter-mean droplet diameter calculation. Since the droplets are solved using fully coupled nonequilibrium gas/particle coupling as implemented in our rocket plume work, the unified breakup rates and vaporization rates control how the vented droplets penetrate into the external stream.

E. Secondary Breakup Methodology

The methodology to analyze how larger droplets breakup into smaller droplets, then vaporize and burn follows directly from that used in modeling spray combustion problems, but the environment is quite different and it varies with both altitude and with venting details. Secondary breakup, at a basic level, is directly related to the Weber number, which is defined as,

$$We = \left(2\rho_g r |U_g - U_p|^2 / \sigma \right) \quad (8)$$

where σ is the surface tension of the liquid composing the droplet. Physically, the Weber number represents the ratio of the aerodynamic inertial force to the surface tension of the droplet. In regions of high shear where the difference in gas and droplet velocity are high, the Weber number will be significant and is an indication of when secondary breakup is likely to occur. As the inertial force grows there is a critical Weber number where the surface tension is no longer sufficient to hold the droplet together and secondary breakup occurs. This critical value is typically taken to be around 12. There are three basic regimes, based on the initial value of the Weber number, which secondary breakup can be classified in terms of. These are described in Table IV.

Table IV. Droplet Breakup Mechanisms

Gas Weber Number	Breakup Mechanism	Description
$12 < We \leq 100$	Bag Breakup	Thin bag forms behind droplet rim
$100 < We \leq 350$	Stripping Breakup	Shear forces strip droplets from liquid ligaments
$We > 350$	Catastrophic Breakup	Droplet immediately disintegrates

Evaluation of the Weber number, as well as the effective saturation pressure as described in the previous section, requires accurate calculation of the droplet surface tension, which is a function of temperature. Although various analytical methods do exist for predicting the surface tension, these methods are complex and no generalized method exists for all liquids. A simpler approach is to create curve fits to experimental data, and this was the approach taken with the CRAFT CFD code. Third order polynomial fits were created for water and a variety of pure hydrocarbon liquids using the NIST WebBook tables which were themselves generated from a variety of experimental sources⁹. The temperature was varied over a range from near the freezing point up to the critical temperature. A comparison of the polynomial fit to the NIST database is shown in Figure 2 for decane, $C_{10}H_{22}$, and it is seen that the polynomial fit does an excellent job of matching the NIST database. This agreement is typical for all the liquids in the plume code database.

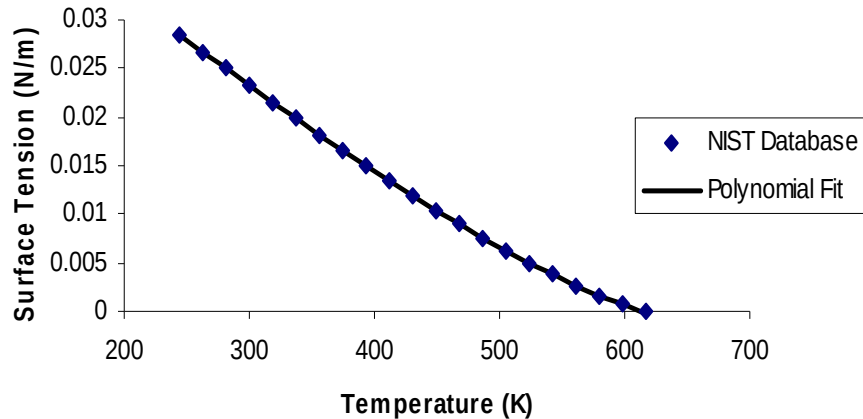


Figure 2. Polynomial Fit for Decane Surface Tension.

IV. Venting and Contrail Studies

A. Gas-Phase Venting Study

The venting of a gas from a missile at lower altitudes can occur when pressure relief valves are used to reduce propellant tank pressures due to cryogenic vaporization or as part of a controlled engine shutdown procedure during the missile staging event. The vented gas may burn due to interactions with the missile aerodynamic flow, and may also enhance the afterburning of the plume. The ability of plume CFD codes to properly analyze this problem largely resides in the modeling of the chemical kinetics. A study was performed at low altitude using CH_4 as a fuel vent gas, which has a long ignition delay and slow chemistry, to evaluate what chemical kinetics are required. This study was conducted on a generic missile body flying at 10 km. The venting of gaseous methane was assumed to come from a 6-inch diameter vent port in the center of the missile. The methane fuel was assumed to be stored cold

(300K), and at a pressure of 2 atm. Table V lists the relevant conditions for the missile, fuel, and plume characteristics.

Table V. Free stream, plume, and Fuel Vent Orifice Reference Conditions

Free stream (O ₂ , N ₂)	Vent Port (CH ₄)	Exhaust Plume
- Alt = 10 km - M = 1.63 - T = 233.1 K - P = 4 psia	Assumed Conditions 6-inch diameter hole - Sonic at hole exit - Cold Fuel - Total Temp 300 K - Total Pressure 2 atm	- M = 3 - T = 1365. K - P = 60000 Pa - Composition (Mass Fraction) - CO 17% - CO₂ 14% - H₂ 1.5% - H₂O 30% - N₂ 37%

To represent the chemical kinetics of both the missile plume and methane being vented, finding a “universal” mechanism representative of both plume afterburning and CH₄ air/plume kinetics needed to be addressed. Plume afterburning was well represented by standardized chemistry mechanisms in plume codes (i.e., System II in SPF), but this was not appropriate for the methane chemistry. An extended chemical kinetic system that was sufficient for methane/air combustion (*AFRL Full Hydrocarbon Mechanism*, without any of the Nitrogen based reactions) was evaluated by using this system for the plume.

The plume only comparisons using the *AFRL Full Hydrocarbon Mechanism* matched very well compared to the well calibrated System II plume chemistry package. Figure 3 shows temperature comparisons for the plume-only cases along with comparisons of H₂ mass fraction. The above simulation was then run using the Full Mechanism with the plume on and with the methane fuel being vented. The results showed that there were obvious signs of the vented fuel burning after being entrained into the hot plume. This is shown in Figure 4, where the top half of the plume is hotter than the lower half.

Further evidence of the vented fuel burning is found by looking at mass fractions of minor product species. CH₃ is present as a product both where the fuel is vented, and at the plume in Figure 5. Presence of some minor product species HCO and CH₂O are shown in Figure 6, and are also indicators of the vented fuel burning with the plume.

Conclusions drawn from this study are that gaseous venting effects can be detected at lower altitudes where the fuel can burn and enhance afterburning, even with a slow burning fuel such as methane. A detailed *Full Hydrocarbon Mechanism* provided an effective means of modeling both the missile plume and fuel being vented from the missile’s fuel tank. Burning was evident as the vented fuel mixed with the hot plume, and, there was a clear increase in temperature where the mixing occurred, along with the presence of product chemical species.

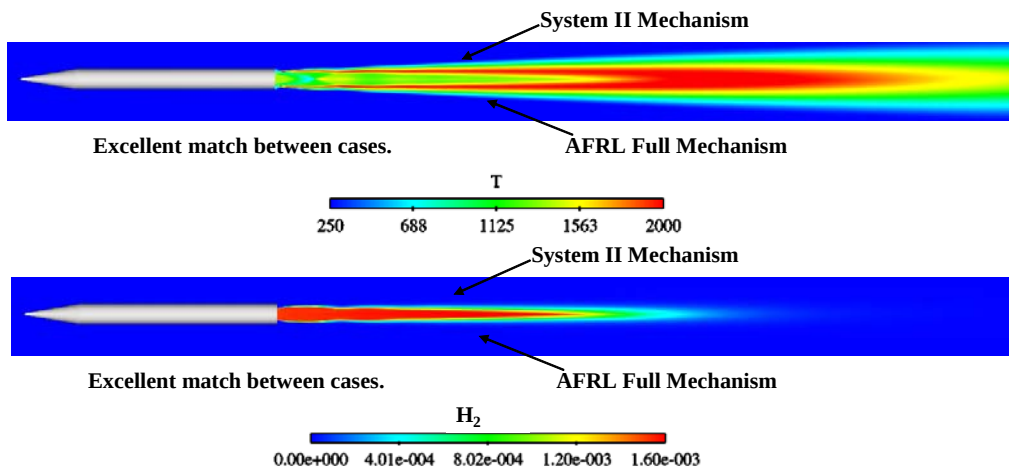


Figure 3. Temperature Contours, and Mass Fraction of H₂ Show Good Agreement for the Full Mechanism With System II Chemistry for the Plume Only Case.

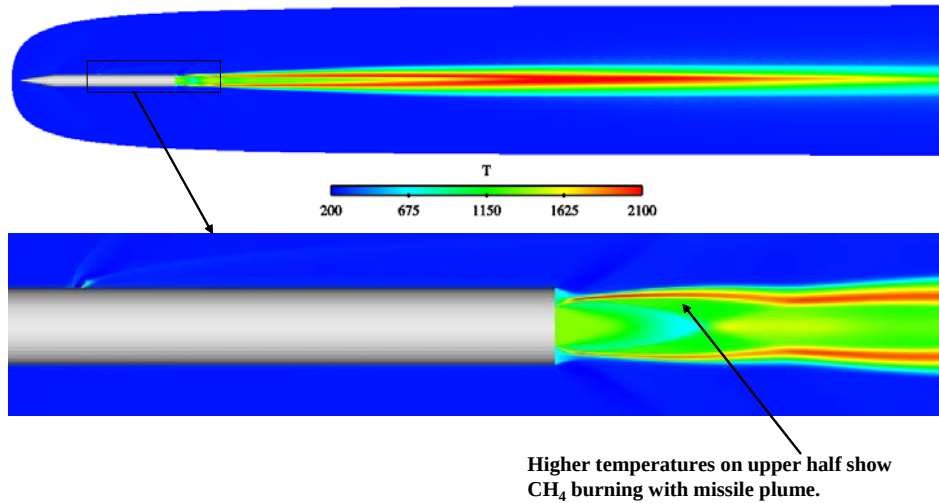


Figure 4. Temperature Contours Show Vented Fuel Burning With Hot Plume.

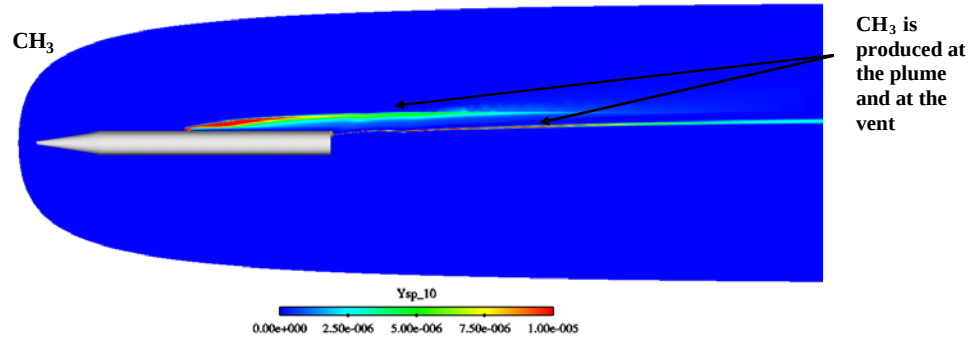


Figure 5. Product Mass Fraction of CH_3 Shows Some Burning as Fuel Exits Vent Port, and as it Mixes With the Plume.

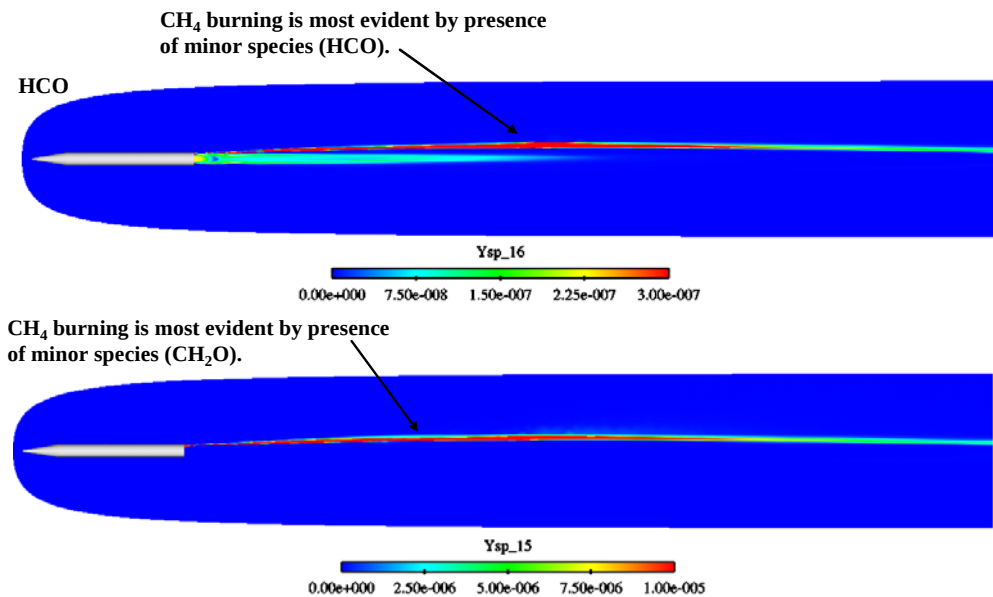


Figure 6. Presence of Minor Species (HCO Mass Fraction and CH_2O mass Fraction) Another Indicator of Vented Fuel Burning.

B. Liquid Venting Study

An important class of problems being studied involves the discharge of a liquid jet into an aerodynamic flow. Computationally, this type of problem involves a different set of challenges compared to gas-gas problem, due to the large variation in fluid density and compressibility between the two phases. Furthermore, in a high shear environment, the liquid stream can breakup into droplets, and based upon local thermodynamic conditions, could vaporize into a gas phase. Thus, a gas-gas problem could ensue, similar to that presented in the previous example. In this case, however, the characteristics of the jet/freestream interaction, such as jet penetration and gas mixing, could be significantly different, depending on the liquid jet breakup. Here, we consider a gas/liquid problem and present the results of a simulation in which the gas/liquid interface is tracked as a first step towards a future complete simulation including liquid jet breakup and vaporization.

The new gas/liquid methodology, discussed in Ref [8], is used to simulate the discharge of a liquid jet into a gas freestream, and track the gas/liquid interface. Two different transverse jet cases are demonstrated with the gas moving (downward in Figure 7) with a velocity of 100 m/s and the liquid jet penetrating the gas stream at a velocity of 5 m/s. Figure 7 presents contours of gas volume fraction at the two different freestream pressures. Figure 8 presents corresponding velocity contours. As expected, the liquid jet penetrates further into the freestream for the lower pressure (1 MPa) case. In either case, the gas/liquid interface is captured, and a relatively high inter-phase velocity is predicted, which provides a mechanism for breakup of the liquid jet. Work is ongoing to predict primary breakup using the model described in Ref. 6.

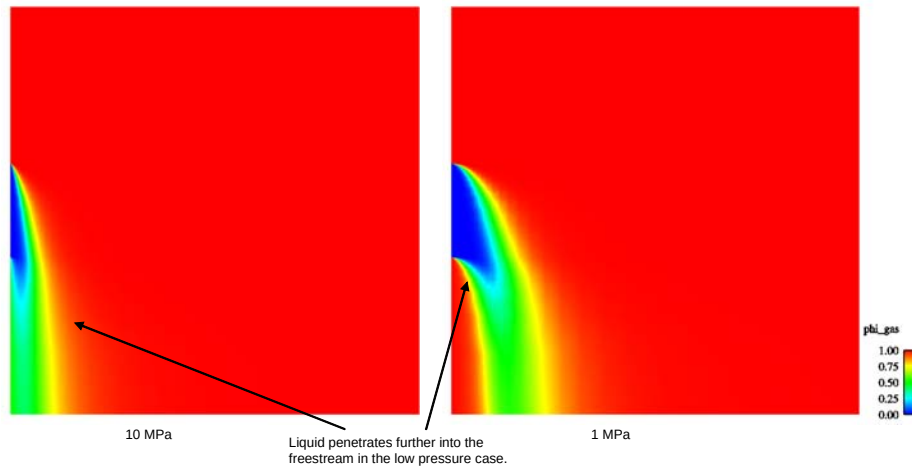


Figure 7. Transverse Liquid Jet; Gas Volume Fraction.

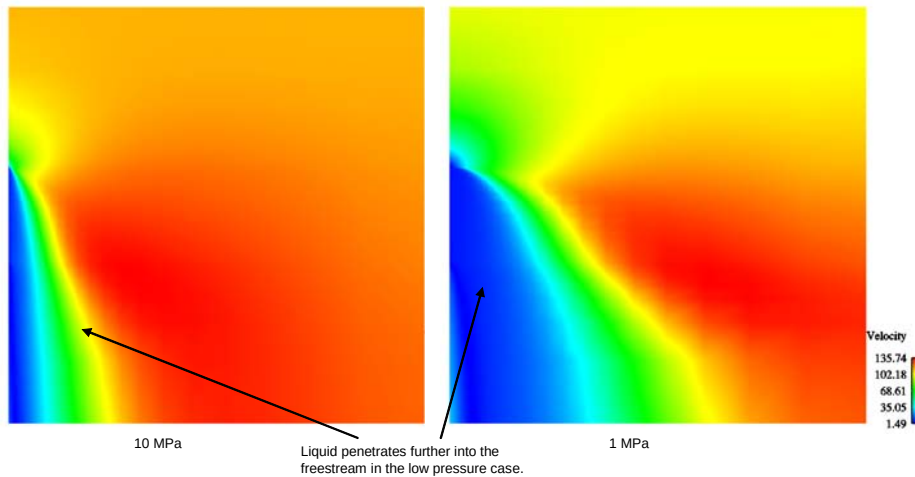


Figure 8. Transverse Liquid Jet; Velocity Contours.

C. Co-Flowing Jet Liquid/Gas Simulation

The simulation of a round water jet surrounded by a high speed annular air jet was performed based on experimental work done by Lasheras et al.¹⁰. Figure 9 shows a schematic of the problem. The simulation was run to match one set of experimental conditions. The inner jet is a 100% liquid core by volume with a nozzle exit velocity of 1.5 m/s. The liquid core is surrounded by an outer air jet with an exit velocity of 250 m/s. The simulations were run at room temperature and pressure (300 K and 1 atm). This experiment also provides a good test case for the primary breakup model that is currently under development. Presented here will be the results of the gas/liquid methodology to capture the interface between the two phases, without breakup. Figure 10 shows the stream-wise velocity contours of the two jets. Figure 11 shows the mass fraction contours of the liquid jet. The gas and liquid begin to mix immediately, and the high shear between the two phases will result in liquid breakup as recorded by Lasheras et al.¹⁰ that will be accounted for in an ongoing study.

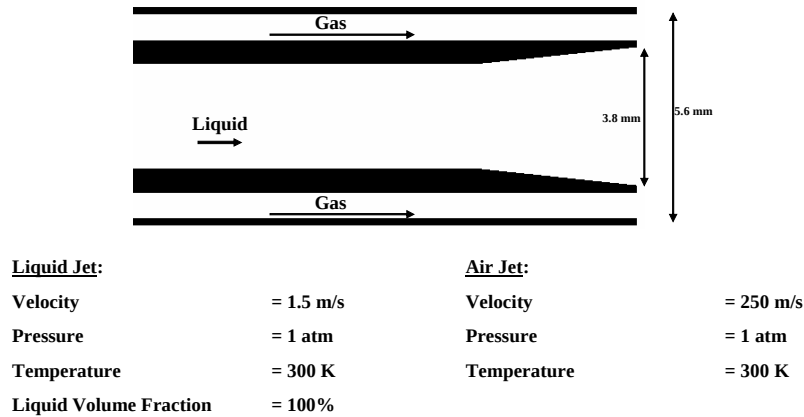


Figure 9. Schematic of Co-Annular Liquid/Gas Jet Problem.

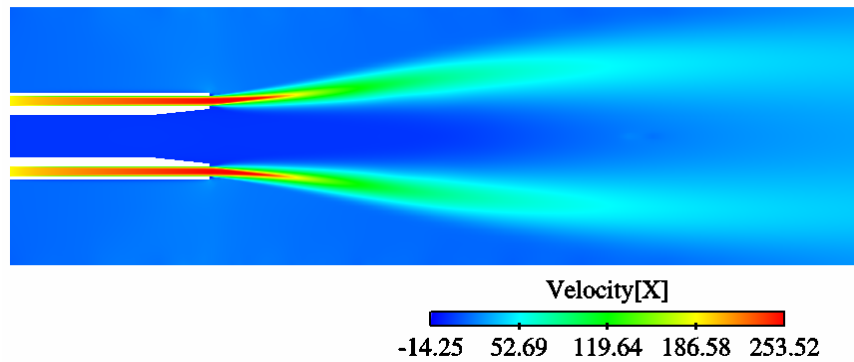


Figure 10. Stream-Wise Velocity Contours.

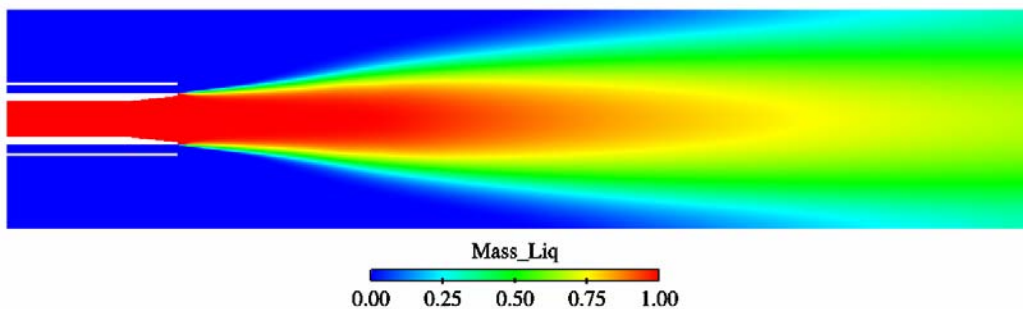


Figure 11. Contours of Liquid Mass Fraction.

D. Unified Secondary Breakup and Droplet Vaporization in a Transverse Jet

The previous example of a liquid jet discharging into a high-speed gas flow has the potential for liquid jet breakup into relatively large droplets, due the primary breakup mechanism, as discussed in Ref. 9. Droplets in a high shear environment are subject to secondary breakup which can result in further disintegration of the droplet, based on the droplet Weber number.

Considered here is the response of pre-existing droplets to the secondary breakup mechanism coupled to droplet vaporization. The simulation compares the penetration of vented 50 micron radius droplets into a high speed stream with (1) no secondary breakup, (2) low Weber number breakup (where a parent droplet breaks into several child droplets, each having about 10% of the radius of the original droplet) and (3) high We number, catastrophic type breakup (where each droplet breaks into numerous droplets, each having a radius that is 2% of the original droplet size). The problem is shown schematically in Figure 12, with Toluene droplets injected into a Mach 2 air freestream. Figure 13 presents the droplet radius behavior (note the difference in scales) for each case, and shows the prominent effect of secondary breakup on droplet size. The left-hand side of Figure 14 shows the droplet cloud penetration, while the right-hand-side shows the conversion of droplets into gaseous fuel due to vaporization. The impact of secondary breakup on droplet vaporization is clearly evident in this unit problem, as the smaller droplets provide a larger surface-area/mass ratio, demonstrating the importance of incorporating a secondary breakup model into droplet vaporization simulations.

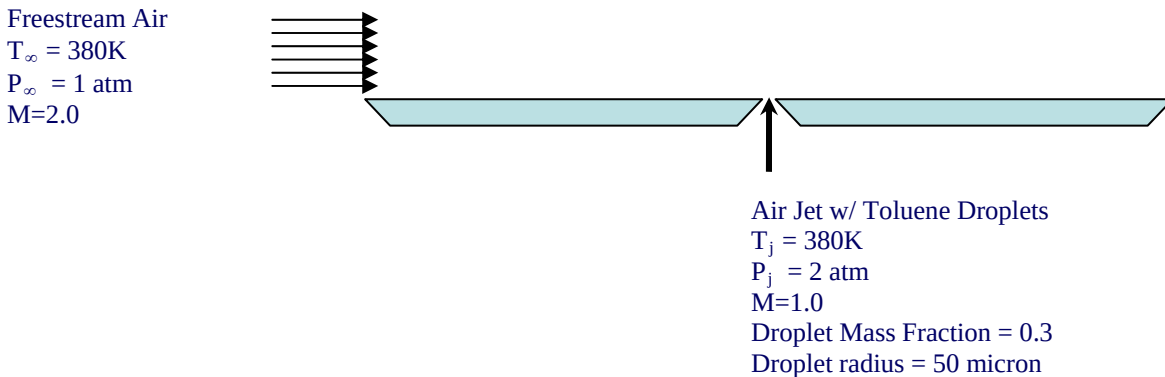


Figure 12. Transverse Jet with Droplet Injection Schematic.

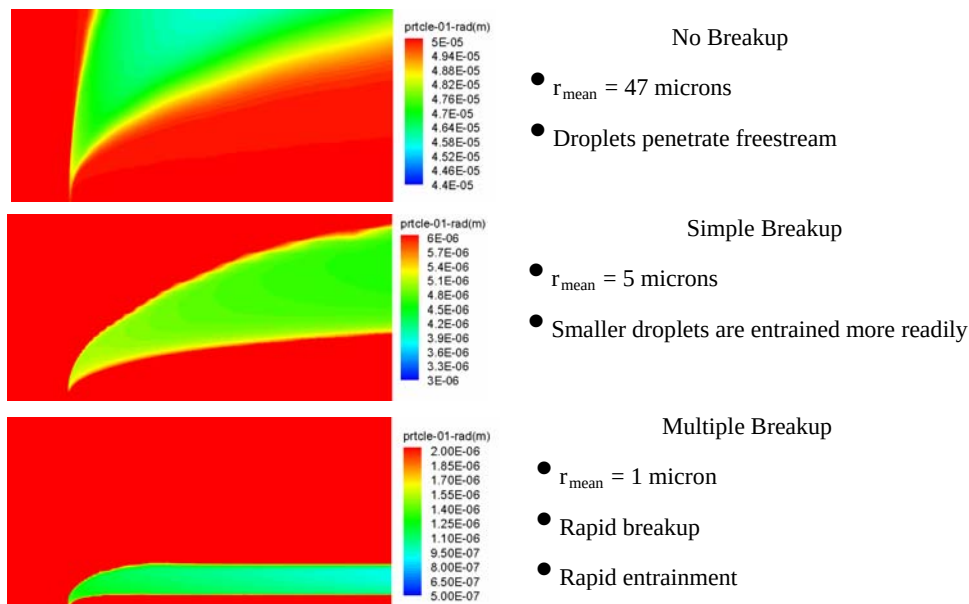


Figure 13. Simple Droplet Venting Example Showing Combined Effects of Secondary Breakup and Vaporization on Droplet Radius.

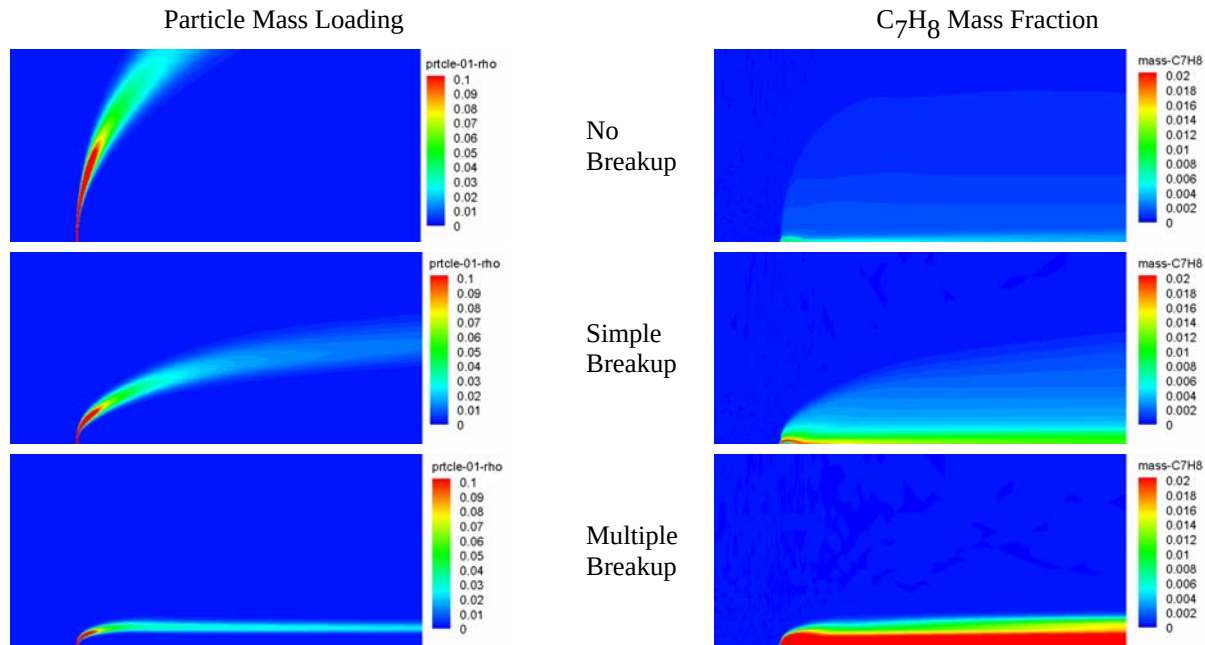


Figure 14. Simple Droplet Venting Example Showing Combined Effects of Secondary Breakup and Vaporization on Droplet Cloud Penetration and Vapor Formation.

E. Contrail Formation Study

Condensation of water vapor in rocket exhaust plumes producing secondary smoke and often extended contrails has proven difficult to model and no well established "practical" model is available in any of the newer CFD plume codes. The approach taken in our work has entailed modifying our Eulerian droplet vaporization/condensation model to consider a multi-component droplet, which consists of a fixed nucleus along with a variable-thickness condensate layer around the nucleus. This is a heterogeneous nucleation model which accounts for the high particle number loading, typical in many rocket exhaust plumes, to provide condensation nuclei. The Sauter-mean droplet diameter model for droplet vaporization is modified to account for the multi-component droplet, and an additional transport equation to track dry nuclei mass distribution is incorporated to facilitate calculation of the composite droplet material properties. The difference between the total droplet cloud density and the nucleus cloud density provides the quantity of condensed vapor at any location in the domain, which reveals itself as secondary smoke in a condensation trail. The ability to predict secondary smoke formation at both low and high altitudes for missile problems is of interest for a variety of logistical reasons. Figure 15 taken from the work of Simmons¹¹ shows imagery associated with sunlight scattered from droplets formed in a 60km plume. Figure 16a (nearfield) and Figure 16b (farfield) show the predicted vapor trail in a 60km plume based on application of the new secondary smoke model.

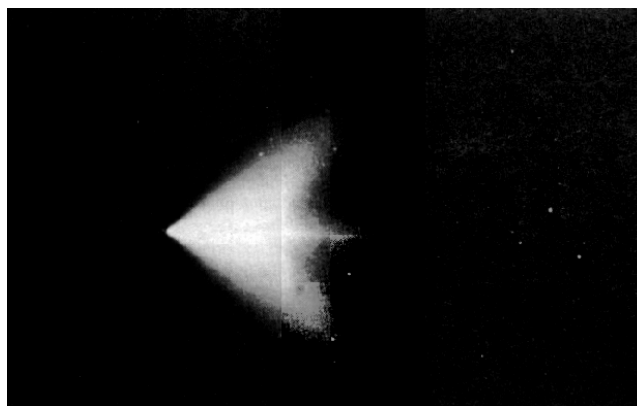


Figure 15. Imagery of Sunlight Scattered from Condensed Water Vapor in 60km Plume (Simmons¹¹)

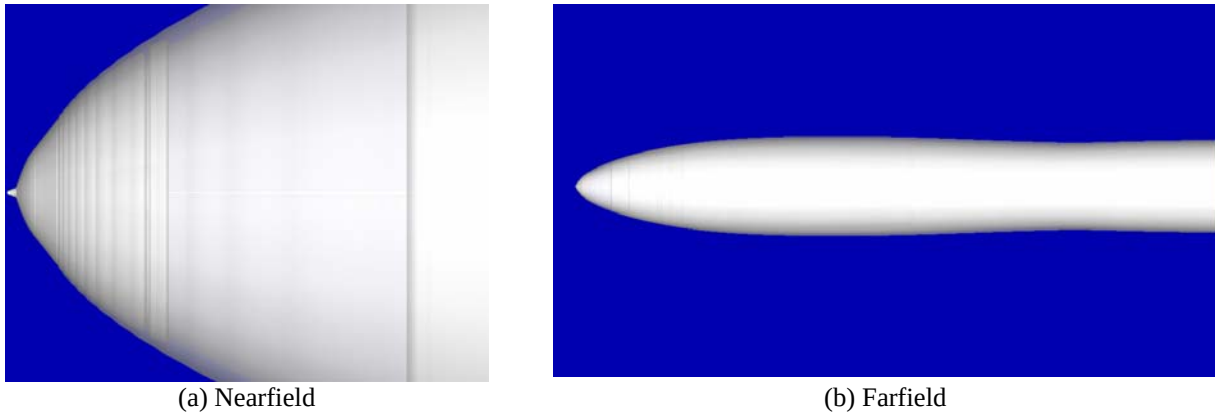


Figure 16. Predicted Water Vapor Formation in 60km Plume using New Secondary Smoke Modeling.

V. Concluding Remarks

Substantial progress has been made in extending the new, engineering-oriented missile plume codes to analyze more generalized problems entailing varied missile propulsive flowfield phenomena. Specifically, engineering models to simulate the complete sequence of liquid jet discharge with breakup, and droplet vaporization/condensation are being implemented to enable the prediction of multi-phase phenomena involved in applications such as fuel venting and plume contrail formation. To analyze such processes, gas/liquid modeling methodology is being incorporated into the plume codes to provide a robust, multiphase interface capturing capability. Overall progress in extending the new plume codes to analyze these processes has been described in this paper and illustrated with a series of example applications.

Acknowledgments

The work described in this paper was performed under an SBIR program entitled "Propulsion Related Missile Phenomena" monitored by AFRL/PRSA.

References

- 1 York, B.J., Lee, R.A., Sinha, N. and Dash, S.M., "Progress In The Simulation of Particulates Interaction In Solid Propellant Rocket Exhaust," AIAA-2001-3590, 37th AIAA/ASME/SAE/ASEE JPC, Salt Lake City, UT, July 8-11, 2001.
- 2 Papp, J.L. and Dash, S.M., "Turbulence Model Unification and Assessment for High-Speed Aeropropulsive Flows," AIAA Paper No. 2001-0880, 39th AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, January 8-11, 2001.
- 3 Calhoon, W.H. and Kenzakowski, D.C., "Flowfield and Radiation Analysis of Missile Exhaust Plumes Using a Turbulent-Chemistry Interaction Model," Paper No. AIAA-2000-3388, 36th AIAA/ASME/SAE/ASEE JPC and Exhibit, Von Braun Civic Center, Huntsville, AL, July 17-19, 2000.
- 4 Sinha, N., Dash, S.M., and Hosangadi, A., "Applications of an Implicit, Upwind NS Code, CRAFT, to Steady/Unsteady Reacting, Multi-Phase Jet/Plume Flowfields," AIAA Paper 92-0837, AIAA 30th Aerospace Sciences Meeting, Reno, NV, January 6-9, 1992.
- 5 Hosangadi, A., Sinha, N., and Dash, S.M., "A Unified Hyperbolic Interface Capturing Scheme for Gas/Liquid Flows," AIAA-97-2081, 13th AIAA CFD Conferences, Snowmass, CO, June 29-July 2, 1997.
- 6 Tonello, N.A., Dash, S.M., and Perrell, E.R., "Advanced Computational Framework For Dynamic Bulk-Liquid Gas Interactions," AIAA Paper-99-2205, 35th AIAA/ASME/SAE/ASEE Joint Propulsion Conference and Exhibit, Los Angeles, CA, June 20-24, 1999.
- 7 Hosangadi, A., and Ahuja, V., "A New Unsteady Model for Dense, Cloud Cavitation in Cryogenic Fluids," Paper No. AIAA-2005-5347, 17th Computational Fluids Dynamic Conference, Toronto, Ontario, CA, Jun. 6-9, 2005.
- 8 Feldman, G., Hosangadi, A., Brinckman, K.W., and Dash, S.M., "Application of Improved Gas/Liquid Methodology to High Speed Venting and Bulk-Flyout Problems," 45th Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan. 8-11, 2007.
- 9 Lemmon, E.W., McLinden, M.O., and Friend, D.G., "Thermophysical Properties of Fluid Systems" In NIST Chemistry WebBook, NIST Standard Reference Database Number 69, Eds. P.J. Linstrom and W.G. Mallard, June 2005, National Institute of Standards and Technology, Gaithersburg MD, 20899 (<http://webbook.nist.gov>).
- 10 Lasheras, J.C., Villermaux, E., and Hopfinger, E.J., "Break-up and atomization of a round water jet by a high-speed annular air jet," *J. Fluid Mech*, Vol. 357, pp. 351-379, 1998.
- 11 Simmons, F.S., "Rocket Exhaust Plume Methodology," AeroSpace Press, El Segundo, CA, 2000.