RECENT THEORETICAL ADVANCES FOR LIQUID-FUEL ATOMIZATION AND BURNING

William A. Sirignano
Department of Mechanical and Aerospace Engineering
University of California, Irvine
Irvine, CA 92697

ABSTRACT

Three modern topics of major importance in the use of liquid fuels for chemical propulsion will be reviewed: atomization of injected streams of liquids, vaporization and burning of large arrays of fuel droplets, and liquid-fuel-film vaporization and burning. The emphasis will be on theoretical developments although some experimental results will be cited. The roles of cavitation and hydrodynamic instability in liquid break-up processes will be analyzed. A new method for treating large numbers of interacting, vaporizing, and burning fuel droplets will be discussed for both unitary and non-unitary Lewis-number cases. The physics of liquid-film vaporization and burning, a new and exciting candidate for use in the miniaturization of combustors, will be analyzed.

NOMENCLATURE

\( a \)  instantaneous droplet radius
\( a_0 \)  initial droplet radius
\( B_H \)  transfer number for heat
\( B_M \)  transfer number for mass
\( c_p \)  specific heat at constant pressure
\( D \)  orifice diameter, mass diffusivity
\( h \)  specific enthalpy
\( K \)  cavitation number
\( L \)  orifice length, latent heat of vaporization
\( N \)  number of species in multicomponent mixture
\( \bar{N} \)  number of droplets in array
\( p \)  pressure
\( \dot{q}_l \)  conductive heat flux per unit area to droplet surface
\( R \)  corner radius of curvature
\( Re \)  Reynolds number
\( U \)  Bernoulli jet velocity
\( \vec{V} \) fluid velocity  
\( We \) Weber number  
\( Y_n \) mass fraction of species \( n \)  
\( \varepsilon_n \) mass flux fraction of species \( n \)  
\( \lambda \) thermal conductivity  
\( \rho \) density

**INTRODUCTION**

The burning of liquid fuels in practical combustors presents many interesting scientific questions. In this paper, an attempt is made to highlight a few current important problems. One issue is the need to understand and to be able to influence the breakup of injected liquids at high pressures where the flow at no point falls below the vapor pressure of the injected fluid. Another issue is the behavior of fuel droplets vaporizing and burning in large groups or arrays where the distance between neighboring droplets is not so large compared to the droplet radius to be able to neglect interactions amongst droplets. Typically, the distance must be more than an order of magnitude larger than the droplet radius which is not satisfied by dense sprays. A third issue addresses the best process for burning liquid fuels in small chamber volumes which actually involves wall filming rather than spraying.

**HIGH-PRESSURE ATOMIZATION: CAVITATION EFFECTS**

The purpose of this study is to identify the role of cavitation in high pressure atomization. The flow in a liquid injector and the resulting jet are analyzed by means of numerical simulation of two-phase flow. This study will lead to a better understanding and predictive capability of effects of cavitation inside the injector on the breakup process. Recent experimental studies\(^1\)\(^-\)\(^4\) show that the occurrence of cavitation inside the nozzle makes a substantial contribution to the breakup of the exiting liquid jet. In the traditional criterion, cavitation occurs when the pressure drops below the breaking strength of liquid \((P_c)\) which in an ideal case is the vapor pressure at local temperature. Winer and Bair\(^5\) and, independently, Joseph\(^6\) proposed that the important parameter in cavitation is the total stress which includes both the pressure and viscous stress. Kottke, Bair and Winer\(^7\) conducted an experiment on cavitation in creeping shear flow where the reduction of hydrodynamic pressure does not occur. However due to high shear stress they observed appearance of cavitation bubbles at pressures much higher than vapor pressure. Archer et al.\(^8\) observed a drop in the shear stress in start-up of steady shearing flow of a low-molecular-weight polystyrene due to opening of bubbles within the flow at stress equal to 0.1MPa.

To study the mechanisms by which cavitation enhances the breakup process, a numerical model is developed using a finite-volume method for integration and a level-set formulation to track the interface and model the surface tension. The
code is validated with several benchmark problems and compared with available literature. The approach requires evaluation of the viscous stress since cavitation depends on molecular-related properties. In particular, averaged values for turbulence cannot be used.

Flow through an Aperture

Flow of a liquid through an aperture is a free-jet problem that offers an analytic solution for the flow field. It has been proved that for a viscous potential flow (VPF), the constant speed condition on the free surface leads to zero normal viscous stress on the free surface, hence satisfies the boundary condition of viscous flow as well. The full Navier-Stokes equations (N-S) for the aperture flow are solved numerically for Reynolds numbers, $Re$, between 1 and 1000 and Weber number, $We$, between 10 and 1000. Employing the theory of viscous potential flow the viscous stresses could be found and using the total-stress criterion for cavitation, the regions that are vulnerable to cavitation are identified and the results are compared to the solution of viscous potential flow. For high Reynolds numbers, solutions are similar except in boundary layers.

Flow through an Axisymmetric Orifice

A numerical simulation of two-phase incompressible flow is conducted in an axisymmetric geometry of the orifice. The orifice has a rounded upstream corner with radius $r$ and a sharp downstream corner with length-to-diameter ratio ($L/D$) between 0.1 and 5. Figure 1 shows an example of solution for velocity field and pressure field for $Re=2000$ and $We=1000$. Cavitation number is defined as $K = \frac{p_u - p_d}{p_d - p_c}$, where $p_u$, $p_d$ are upstream and downstream pressures, and $p_c$ is the critical pressure. In our non-dimensional formulation, the upstream and downstream pressure values, the critical pressure, and therefore the $K$ value are not required to obtain the Navier-Stokes solution. The total stress including viscous stress and pressure has been calculated in the flow field and, from there, the maximum principal stress is found ($T_{11}$). The total-stress criterion for cavitation is applied to find the regions where cavitation is likely to occur and compared with those of the traditional pressure criterion. Results are shown in Figure 2. For each problem, $K$ has a value representative of that problem, say $K=K^*$. In Figure 2, the volume enclosed by the $K^*$ contour is the volume where the cavitation criterion is met and the flow is therefore vulnerable to cavitation. It is observed that the viscous stress has significant effects on cavitation especially for nozzles with larger length-to-diameter ratio. Namely, for any given value of $K$, the volume susceptible to cavitation is larger under the total-stress criterion. As explained in the text, the method is not restricted to any value of $K$. The figure only portrays a sample of the infinite number of contours that can be obtained by post-processing of the Navier-Stokes solutions.
Figure 3a shows the threshold value of cavitation number above which cavitation occurs in the nozzle. The effect of geometry and occurrence of hydraulic flip in the orifice on the total stress is also studied. Here flow in nozzles with different radii of curvature at the inlet corner is considered. $r/D$ is varied between 0.01 and 0.04 while keeping other parameters of the flow and domain constant. Figures 3b and 3c show the threshold value of $K$ versus $r/D$. For both Reynolds numbers of 1000 and 2000, the $K_{th}$ increases as the $r/D$ increases. This is expected because the larger the radius of curvature, the smaller the increase in velocity and drop in pressure, so the less chance of cavitation.

![Figure 1](image1.png)

Figure 1. Velocity profiles and contours of $(p-p_d) \frac{1}{2 \rho U^2}$ for flow with $Re=2000$

![Figure 2](image2.png)

Figure 2. Cavitation threshold with $T_{ij} + p_c = 0$ with $K = 2; 5; 10; Re=1000$, $We=1000; L/D=1$, (a) total stress criterion, (b) pressure criterion.
Figure 3. Threshold values of $K$ above which orifice cavitation occurs; $L/D=2$.

**Laminar Jet Breakup**

The flow of a liquid through a nozzle and resulting jet in a stagnant gas is investigated numerically. The flow is considered to be laminar and axisymmetric. Creation and growth of surface waves due to Kelvin-Helmholtz and capillary instabilities leading to breakup of jet are captured as indicated in Figure 4. The disturbance in jet is caused by perturbing the mass flux of liquid through nozzle. Effects of the geometry such as length-to-diameter ratio and curvature of corners of the nozzle on the growth of interfacial waves and breakup distance are studied. It is observed that longer nozzles create a more stable jet with longer breakup distances. Further calculation will address higher values of $Re$ and $We$. 
Bubbles in the Flow

The growth and collapse of cavitation bubbles in the nozzle flow is simulated by a one-way-interaction model. That is, the liquid flow affects bubble growth and collapse but the bubble size change does not modify the liquid flow in this preliminary calculation. We expect that a future calculation with a two-way-interaction will result in moderation of the maximum bubble size significantly. This analysis provides the effects of different flow parameters such as Reynolds number, Weber numbers and size of nucleation sites on the cavitation character.

In the one-way interaction it is assumed that a nucleation site which could be represented as a sub-micron size bubble is moving with the local velocity of flow and as it reaches the low pressure regions, near corners, it expands and creates a cavitation bubble and later by moving to higher pressure regions, it will collapse. Also, it is assumed the existence of the bubble will not significantly change the flow field around it. In the future, this model will be expanded to consider a two-way-interaction between bubbles and the flow, where the change in the flow field due to bubble growth and collapse will also be modeled.

Starting from a point in the flow, one can find the streamline and position of particle with time and pressure felt by particle. Then, this pressure is used to solve the equation governing size of the bubble. Figure 5 shows the pressure and bubble size in a flow with $Re = 1000$. In the low pressure region near the corner, bubbles grow and then collapse.

The bubble first grows followed by collapse in size with some oscillation. A lag occurs in the bubble size variation; i.e., maximum size does not occur at minimum pressure because capillary pressure and viscous normal stress are considered in the balance at the bubble / liquid interface. Also, phase change is considered as the bubble remains saturated with vapor of the liquid in the isothermal flow.
BURNING OF LARGE ARRAYS OF FUEL DROPLETS

In previous papers, a generalized approach was discussed for liquid-fuel vaporization burning in general configurations, including dense droplet arrays and liquid films. Those works built upon the works by Labowsky, Umemura et al., and Brzustowski and co-workers. Labowsky introduced the mass-flux potential function (which we apply later), first transformed the field equation to Laplace's equation, and examined up to nine droplets. He used the method of images. In a series of papers, Imaoka and Sirignano explored and compared different computational approaches, extended the capability to handle transient as well as quasi-steady cases, calculated arrays as large as 1000 droplets, considered effects of non-uniform initial droplet size and initial spacing amongst droplets, demonstrated the proper procedure for the averaging of the gas-phase transport properties, and obtained correlations based on similarity variables that can be used in lieu of detailed calculations in the future.

Except for Sirignano, the above researches were limited to cases with unitary values of gas-phase Lewis number. Sirignano recently analyzed the case of many droplets and liquid films with Fickian diffusion and non-unitary Lewis numbers. The mass diffusivities differed according to the diffusing species and varied with the spatial location.

The analysis here will apply to very general configuration that includes an array of droplets, liquid wall films, and liquid free films. Variable physical properties will be allowed. If combustion occurs, the chemical kinetic rate is infinite and flames have zero thickness. We will focus here on situations with only Stefan convection and without forced or natural convection. Stefan convection is the flow caused by the vaporized gas moving away from the surfaces of the droplets. We assume that the Stefan velocity at the surface is sufficiently strong to cause the gas velocity to be normal to the liquid-gas interface at that surface. Chemical reactions will not be considered. There are $N$ species in the gas phase,
including the vapors of the N-2 miscible components in the liquid phase, all of
which produce vapors. In addition, oxygen and nitrogen will be present in the
gas with nitrogen concentration dominant amongst the gas species. The Lewis
number will be allowed to take any value.

The gas phase is quasi-steady and, for the case with only Stefan convection, a
mass-flux potential has been shown to exist. So, we find using conventional
omenclature that

$$\nabla \cdot (\rho \vec{V}) = 0 \quad ; \quad \rho \vec{V} = \nabla \phi \quad ; \quad \nabla^2 \phi = 0 \hspace{1cm} (1a, b, c)$$

The mixture enthalpy and the specific heat are mass-flux weighted (rather than
the conventional mass weighting) using the mass-flux fraction.

$$h_e = \sum_{n=1}^{N+P} e_n h_n = \sum_{n=1}^{N+P} e_n \int_{T_{ref}}^T c_{p,n} dT' = \int_{T_{ref}}^T (\sum_{n=1}^{N+P} e_n c_{p,n}) dT' = \int_{T_{ref}}^T c_{p,e} dT' \hspace{1cm} (2)$$

Under these conditions, it has been shown that

$$\rho \vec{V} = \nabla \phi = \frac{\lambda \nabla T}{h_e - H} = \frac{\lambda / c_{p,e}}{h_e - H} \nabla h_e = \rho D_n \nabla Y_n \hspace{1cm} n = 1, ..., N \hspace{1cm} (3)$$

where \( H = \sum_{n=1}^N e_n h_{n,S} - L_{eff} \) and \( L_{eff} = L + \dot{q}_f / (\rho \overline{V}^2) \) \( S \) denotes a gas
property at the droplet surface.

The scalar properties may now be viewed as one-dimensional in the sense that
they can be cast as functions of the potential function only. Integration and
creation of a normalized potential function \( \Phi = \phi / \phi_\infty \) results in the following:

$$\phi_\infty = \overline{\rho D_n \ln[\frac{e_n}{e_n - Y_{n,S}}]} = \overline{\rho D_n \ln[1 + B_{M,n}]} \hspace{1cm} (4)$$

$$\nabla^2 \Phi = 0 \quad ; \quad \Phi_S = 0 \quad ; \quad \Phi_\infty = 1 \hspace{1cm} (5)$$

where the definitions are used that

$$B_{M,n} = \frac{Y_{n,S}}{e_n - Y_{n,S}} \hspace{1cm} ; \hspace{1cm} B_{H,e} = \frac{\sum_{n=1}^{N-2} e_n [h_{n,\infty} - h_{n,S}]}{L_{eff}} = \frac{[h_{n,\infty,e} - h_{n,S}]}{L_{eff}} \hspace{1cm} (6a, b)$$

Note that the mass-flux potential is obtained via solution of Laplace’s equation.
The theory shows that the transport properties must be properly averaged.
The overbar implies the average over the gas surrounding the droplets.

$$\overline{\lambda / c_{p,e}} = \overline{\frac{1}{0} (c_{p,e} / \lambda) d\Phi'}^{-1} \hspace{1cm} ; \hspace{1cm} \overline{\rho D_n} = \overline{\frac{1}{0} d\Phi' / (\rho D_n)}^{-1} \hspace{1cm} (7a, b)$$
Solutions are obtained for the enthalpy and mass fractions using the mass-flux potential as the independent variable.

\[
\frac{Y_n - \varepsilon_n}{Y_{n,S} - \varepsilon_n} = \left[1 + B_{M,n}\right] \frac{\rho D_e \Phi}{\partial \rho D_e} \int \Phi' \left(\lambda\epsilon_{p,x}\right) \frac{0}{\partial} \left(1 + B_{H,n}\right) \int \Phi' \left(\lambda\epsilon_{p,x}\right) 
\]

\[
\frac{h_e - h_{e,S} + L_{eff}}{L_{eff}} = e^0 = e = [1 + B_{H,n}] \int \Phi' \left(\lambda\epsilon_{p,x}\right) 
\]

\[
\frac{h_e - h_{e,S} + L_{eff}}{L_{eff}} = e = [1 + B_{H,n}] \int \Phi' \left(\lambda\epsilon_{p,x}\right) 
\]

Phase equilibrium at the liquid-gas interfaces are established through the use of Raoult’s law with the Clausius-Clapeyron relation. Balances of energy and mass between the phases at the interfaces are imposed and the unsteady diffusion equations for energy and mass are solved in the liquid interior.

Vaporization rates have been calculated for a wide range of droplet-array configurations13, 15 with up to 1000 droplets. It has been shown that a similarity solution can be correlated with the numerical results. In particular, we define \(\eta_A\) as the ratio of the droplet mass vaporization rate (averaged over all droplets in the array) to the mass vaporization rate for an isolated droplet. \(\xi\) is the similarity variable which depends on the array volume \(V_A\), the total liquid volume in the array \(V_l\) and the number of droplets in the array \(N\). Specifically, the computational correlation yields that

\[
\eta_A = 1 - \frac{1}{0.725671 \varepsilon^{0.97176}} ; \quad \xi = \frac{\left[\frac{4\pi V_A N}{3V_l}\right]^{1/3}}{\left(N^{1/3} - 1\right)^{0.72}} 
\]

Note that, for large values of the number of droplets in the array, the similarity parameter \(\xi\) is approximately the droplet spacing-to-radius value divided by \(N^{0.72}\) and as the value of \(\xi\) increases, the average vaporization rate gets closer to the isolated droplet value. It is found that as the ratio of droplet spacing \(d\) to initial droplet size \(a_o\) decreases, the plot of the square of instantaneous radius \(a\) versus time becomes more convex, deviating increasingly from a straight line. The dimensionless time is normalized by the lifetime of an isolated droplet of the same initial size. The results, as shown for a nine-droplet cubical array in Figure 6, indicate that the interior droplets in an array vaporize more slowly
than the droplets on the outer edge of the array. A decrease in the spacing ratio causes a decrease in vaporization rate.

Flame location is determined using the standard thin-flame condition; i.e., at the flame, oxidizer and fuel mass fractions are zero. The distance from the droplet surface to the flame location increases with liquid volatility, i.e., increases with increasing surface temperature, decreasing latent heat, and decreasing boiling point. Figure 7 shows the result for a five-droplet array. The flames for the less volatile methanol surround individual droplets while an envelope flame exists for decane. Figure 8 shows for a four-droplet array how the flame distance increases as the surface temperature increases. Of course, with transient heating the droplet surface temperature will increase with time. The result of increasing surface temperature is that a bifurcation occurs with a transition from individual flames around each droplet to an envelope flame.

The more volatile component in the liquid droplet will vaporize earlier and faster than other components. The surface regresses as vaporization occurs to create an effective advection towards the surface; in addition, the more volatile component diffuses towards the surface where its concentration has been decreased due to vaporization. The least volatile component will diffuse away from the surface towards the droplet center because its slower vaporization results in an increase in its composition near the surface. Figure 9 shows a three-component liquid which initially has a two-thirds mass-fraction value for heptane and one-sixth value for each of octane and decane. The dimensionless radial position within the droplet is normalized by the instantaneous droplet radius. Near the surface, preferential vaporization leads to lower values of heptane and octane and a higher value of decane.23

Figure 6. Nine-droplet cubical array, eight outer droplets, one centered droplet.
Figure 7. Flame location in five-drop array. Methanol, isolated, individual flames; decane, envelope flame.

Figure 8. Four-decane-droplet array with flame location and surface temperature. The flame moves away from the droplet surface as surface temperature increases.
LIQUID-FILM COMBUSTOR

A new concept\textsuperscript{24, 25} for miniature combustors is to inject all or a portion of the liquid fuel directly as a film on the wall surfaces. On those walls, high heat transfer from the combustion products normally occurs. The film burning described herein can be applied to rockets, ramjets, turbojets, (reciprocating and rotary) internal combustion engines, heating furnaces, kilns, boilers, and to any other combustor where heat losses must be reduced and/or quenching must be prevented. Various geometrical combustor shapes can be used for the continuous or intermittent combustion application of this concept. Here, we shall discuss the application to continuous cylindrical combustors.

Current technology for larger combustors does not rely on any significant portion of the liquid fuel filming on the combustor walls. Rather, to keep the ratio of liquid surface area to liquid volume large enough to sustain the fuel vaporization rate at the large required value, the fuel is injected as a spray. The intention is to vaporize the liquid as a spray before very much liquid deposits on the walls or solid surfaces of the combustor. That is, the droplets are suspended in the gas or at least the gas affects their motion so that they do not reach a solid surface and adhere to it before vaporization is completed. If the fuel were filmed in these larger engines, the surface area of the liquid would not be large enough to sustain the needed vaporization rate. However, the surface-to-volume ratio of the combustor will grow as the volume of the combustors decreases. As a consequence, in the sub-centimeter combustor-scale range under discussion here, the liquid film can offer as high a liquid surface area for vaporization as a vaporizing spray can. Film vaporization rates in miniature
combustors will be adequate compared to spray vaporization rates on account
of the large surface area per unit volume. So, film combustion is as competitive
as spray combustion in miniature combustors from this perspective. Film
vaporization would not be competitive, of course, in larger combustors. Film
vaporization/combustion gains over spray combustion in terms of limiting wall
heat losses that can cause a reduction in efficiency and a potential for flame
quenching. With a spray, we have heat transferred from the gas to the walls
plus from the gas to the droplets. Energy transferred to the liquid is regained by
the gas phase upon vaporization but wall losses are real losses. With the wall
film, the heat cannot be transferred to the wall directly but rather goes to the
liquid. Nearly all of this energy returns to the gas phase upon vaporization. The
wall receives very little heat passed through the liquid and never achieves a
temperature above the liquid boiling point. These temperatures are substantially
lower than current technology would allow without some other mechanism for
cooling the walls.

The first laboratory demonstrations were done at UCI. A schematic of the
experiment is shown in Figure 10 and an image from the experiment is shown
in Figure 11. The pyrex-glass cylindrical combustor that was fabricated with an
internal diameter of one centimeter is seen in Figure 11. The air flows past
swirl vanes downstream into the combustion chamber where liquid fuel is
introduced tangentially through eight one-millimeter-diameter feed tubes, a pair
at each of four axial positions. The transparent combustor allowed
determination of the fact that the liquid does form a film as it flows over the
inside cylindrical surface. It was difficult to ignite the methanol liquid-fueled
flame from a cold start, so both methane gas and liquid methanol fuels were
used simultaneously. (With heptane liquid fuel, the gas assist was not
necessary.) The procedure was to feed a bit of liquid fuel into the base of the
combustor, flow the fuel/air mixture and ignite the flame at the exit of the tube.
The flame was now fed by both some liquid picked up as the air flowed past the
pool at the base of the combustor and by the gas. We would then decrease the
gas fuel flow rate until at a critical condition, the flame jumped into the tube,
where it burned in a confined state, as shown in Figure 10. It is important to
reiterate that burning occurred inside the tube only when liquid fuel was
provided. In fact, if the liquid flow was stopped leaving only the inflow of gaseous fuel, the flame continued to burn inside the tube until it exhausted any remaining film and it then jumped up to the top of the tube to burn in the low-swirl stabilized state. Other experiments\textsuperscript{24} have examined improved methods of swirl to mix the fuel vapor and air faster and allow shorter combustors.

Figure 11. Confinement of flame with methanol liquid wall film. One-centimeter-diameter pyrex tube with tangential fuel injection at several locations. Swirler inflow of air at bottom and exit of products at top. The flame extends beyond the exit.

A theoretical effort\textsuperscript{26} was performed using a linearized approach to the advection/ diffusion/ reaction equations. The model does not consider the application of the film but assumes that the wall is lined with a liquid fuel film. Air flowing into a cylindrical chamber with a thin film of fuel on the walls was considered. Vaporization, mass diffusion, heat diffusion, and fast oxidation kinetics were modeled. The flame position for heptane fuel at a bulk temperature of 298K was calculated at three different Peclet numbers under the assumption of infinite chemical kinetic rate so that a flame of zero thickness results.

As expected, the flame moves further downstream with increasing Peclet number. At $Pe = 500$, the combustion is completed at approximately 12.5 diameters length as indicated by the flame collapsing to a point on the axis. If relatively high mass flows and relatively short chamber lengths are desired, it is clear that the effective diffusivity must be increased through the generation of vortices or turbulence.
Temperature and mass fraction profiles for heptane are presented in Figures 12 and 13. Peak temperature occurs at the flame with fuel vapor and oxidizer (air) each existing on only one side of the flame with both mass fractions decreasing with radial distance from the flame and going to zero values at the flame. The heptane case leads to faster vaporization and shorter flame length, while a change in the initial fuel temperature has little effect. Figure 14 shows an important implication of the integrated vaporization rate. $L_{\text{flame}}$ is the downstream position at which combustion is completed; that is, it is value of the $x$-position where the flame crosses the axis of symmetry. It is seen to increase roughly linearly with increasing Pecllet number. Essentially, an increase in air velocity or a decrease in diffusivity will elongate the flame. Furthermore, a greater distance is required for the less volatile fuel, methanol, than for heptane. $L^*$ is the fuel film length required to vaporize at an integrated rate that matches, in stoichiometric proportion, the inflow of air. It is seen in Figure 14 that $L^* < L_{\text{flame}}$ for $Pe = 10$ or greater.

A careful examination of the data indicates that the dependence of $L^*$ on $Pe$ is sub-linear; that result is qualitatively in agreement with boundary layer theory which predicts a square-root dependence. The implication of the linear and sub-linear dependencies is that a crossing of the curves will occur but, apparently, it happens for a very small $Pe$ value. The result implies that, in order to protect the chamber wall for the full length needed to complete combustion, the fuel must be supplied in excess of stoichiometric mixtures. An alternative option not yet explored is to mix the fuel with a less volatile inert liquid (e.g., alcohol and water) to get the wall coverage for heat protection without exceeding stoichiometric proportions significantly. Or the inert liquid could be applied to the chamber walls downstream of the end of the fuel film. Also, the addition of turbulence or vorticity in the gas can create a larger value of the effective diffusivity. Qualitatively, this is similar to decreasing the value of $Pe$ implying that the increased transport rates will shorten the required lengths for completion of vaporization and combustion.

**CONCLUDING REMARKS**

Results from theoretical and computational analyses have been presented for some of the forefront problems in the burning of liquid fuels. Explanations for the cause of cavitation and the bubble-growth-and-collapse history in high pressure injector systems have been provided. The influence of neighboring droplets on the vaporization and burning of a fuel droplet in a dense spray has been analyzed. Modified droplet heating, vaporization, and burning laws have been developed. A new strategy for vaporizing and burning liquid fuel in miniature combustors has been discussed. For the liquid-film combustor, the feasibility of the concept and the need to improve mixing rates have been demonstrated.
REFERENCES


Figure 12. Calculated temperature contour in axisymmetric heptane/air liquid-film combustor. Air flows from left to right. Peclet number =500. Parabolic axial-velocity profile.
Figure 13. Calculated heptane fuel-vapor and oxygen mass fractions contours in axisymmetric liquid-film combustor. Air flows from left to right. Lengths are normalized by cylinder radius. Peclet number = 500. Parabolic axial-velocity profile.

Figure 14. Film length for stoichiometric flux proportion and flame length versus Peclet number for methanol and heptane fuels. Plug-flow Axial-velocity profile.