

Available online at www.sciencedirect.com



International Journal of Heat and Fluid Flow 26 (2005) 656-671

Thermational Journal of HEAT AND FLUID FLOW

www.elsevier.com/locate/ijhff

# Pulsed, supersonic fuel jets—A review of their characteristics and potential for fuel injection

B.E. Milton<sup>a,\*</sup>, K. Pianthong<sup>b</sup>

<sup>a</sup> School of Mechanical and Manufacturing Engineering, University of NSW, Sydney, NSW 2052, Australia <sup>b</sup> Department of Mechanical Engineering, Ubon Ratchathani University, Warinchamrab, Ubon Ratchathani 34190, Thailand

> Accepted 1 March 2005 Available online 1 June 2005

#### Abstract

High pressure fuel injection has provided considerable benefits for diesel engines, substantially reducing smoke levels while increasing efficiency. Current maximum pressures provide jets that are at less than the sonic velocity of the compressed air in the cylinders at injection. It has been postulated that a further increase into the supersonic range may benefit the combustion process due to increased aerodynamic atomization and the presence of jet bow shock waves that provide higher temperatures around the fuel. Pulsed, supersonic injection may also be beneficial for scramjet engines. The current program is examining pulsed, supersonic jets from a fundamental viewpoint both experimentally and numerically. Shock wave structures have been viewed for jets ranging from 600 to 2400 m/s, velocity attenuation and penetration distance measured, different nozzle designs examined and autoignition experiments carried out. Inside the nozzle, numerical simulation using the Autodyne code has been used to support an analytic approach while in the spray, the FLUENT code has been used. While benefits have not yet been defined, it appears that some earlier claims regarding autoignition at atmospheric conditions were optimistic but that increased evaporation and mixing are probable. The higher jet velocities are likely to mean that wall interactions are increased and hence matching such injectors to engine size and airflow patterns will be important.

© 2005 Elsevier Inc. All rights reserved.

Keywords: Supersonic injection; High pressure injection; Advanced diesel injection

#### 1. Introduction

Fuel injection and the subsequent spray formation is a key element in the technology of all internal combustion engines, these being gas turbines, spark ignition and compression ignition (diesel) engines. The last is the most demanding because:

• a separate injection event of short duration occurs every second cycle (i.e. at half the engine speed) in a typical four stroke engine;

- the injection timing controls the combustion initiation and hence must be precise;
- the fuel and air mass flow rates are not in a fixed ratio and hence an accurate injection duration related to the injection pressure is necessary to provide the correct fuel quantity at each injection for the particular speed/load operational point.

Diesel engines play an increasingly significant role in many aspects of modern society dominating the heavy road transport, agriculture, mining and marine sectors and being of considerable importance in rail transport, power generation and pumping. With increasing

<sup>\*</sup> Corresponding author. Fax: +61 2 9663 1222. *E-mail address:* b.milton@unsw.edu.au (B.E. Milton).

<sup>•</sup> the injection must be of very high pressure as it is into the cylinder towards the end of compression. Note that diesel compression ratios are very high;

<sup>0142-727</sup>X/\$ - see front matter @ 2005 Elsevier Inc. All rights reserved. doi:10.1016/j.ijheatfluidflow.2005.03.002

legislative control over engine emissions and the requirement, often contradictory, for improved fuel efficiency, there have been many changes in recent years. One of the most important developments has been in the area of fuel injection. This is not new as Rudolph Diesel himself undertook extensive research into air blast type atomisers during the original development of his engine in the 1890s (see Cummins, 1993). However, for a long period after the introduction of single fluid systems (i.e. fuel only, often termed anomalously "solid injection"), mechanical injection dominated from the first to the last decade of the 20th century. This included distributor pumps, in-line pumps and unit injectors with progressively higher pressures although, until recently, the last were used only on very large engines. Then, as the need to control particulate matter (PM), of which exhaust smoke is a large component, and nitrogen oxides  $(NO_x)$  simultaneously, new types of systems were introduced. These were the mechanical/solenoid controlled electronic unit injectors (EUI), the constant high pressure pump/solenoid controlled common rail types (CRI) and the hydraulically actuated, electronically controlled unit injector (HEUI) with a constant, medium pressure oil pump and an amplifying piston within the injector. They essentially allow both very high pressure injection to provide improved fuel atomization and control of the fuel timing and delivery from the engine management system. Typical data on the maximum pressures for various systems currently in use are

- Distributor and in-line pumps, 75 and 110 MPa respectively.
- Mechanical and EUI unit injectors, 100-200 MPa.
- Common rail injectors, 140–180 MPa.
- HEUI types 160–230 MPa.

It can be seen that the maximum injection pressures (i.e. those in the nozzle sac) of the different types range from about 75 to 230 MPa in systems currently or shortly to be in production. For a small nozzle diameter and intermittent injection such as that in a diesel engine, a coefficient of velocity,  $C_v$ , is likely to be about 0.6–0.7 and the corresponding jet velocity immediately at the nozzle exit is in the range between 255 and 520 m/s. Typically, the air in a diesel engine during the injection process is at about 750 K and hence has an acoustic velocity of about 550 m/s. Hence, the Mach number of the jet lies in the subsonic range of approximately M = 0.45-0.95.

Further increases in injection pressures will render the jet velocity supersonic with a consequent alteration to the external flow. As with supersonic solid bodies, leading edge shock waves will eventuate with consequential modification of the shape of the spray and an increase in the local air temperature due to the shock entropy increase. Surface wave phenomena that influence the initial jet breakup will become more significant and the higher velocities will increase the shear-induced atomization. Overall, further enhancement of the jet atomization, evaporation and mixing processes and a reduction in the ignition delay period are possible. It has been postulated (Field and Lesser, 1977; Shi, 1994; Shi and Takayama, 1999) that fuel jets of sufficiently high supersonic Mach numbers may autoignite spontaneously in air at atmospheric conditions.

While diesel engines may benefit from such fuel jets in the low supersonic range, the higher Mach number jets may have significance in other applications. One of these is in scramjet (supersonic combustion ram jet) engines that are being considered for suborbital flight. These engines rely on the strong oblique shocks formed at the intake in hypersonic flight to provide sufficient air conditions for the combustion to take place by direct fuel injection into the very high velocity air stream. As these are oblique shocks, the air flow is still supersonic and thus combustion must occur in a very fast flow. For this to be practical, the combustion must be completed within the residence time associated with the combustion chamber. This is extremely short and places huge demands on the physical and chemical processes involved in the spray atomization, fuel evaporation, mixing and ignition. While hydrogen has been used in most experiments, there are significant advantages in using conventional liquid fuels for energy density and storage purposes but their mixing and evaporation is relatively slow. Intermittent jets have significant advantages over continuous ones in mixing rates. A subsonic intermittent jet injected into a supersonic air stream (Xu et al., 1999) is shown in Fig. 1. Here, the shock waves are induced by the interaction of the supersonic air and subsonic liquid and help in deforming and breaking up the jet. A supersonic jet with its own shock pattern would result in even more complex shock wave structures due to the interactions and a further increase in atomization rates is likely. Appropriate experiments need to be considered.

To date, there has been only limited research on intermittent, supersonic jets. Much of this has been directed towards jet cutting and cleaning processes that require a coherent jet (Jenkins, 1955; Bowden and Brunton, 1958, 1961). Atomization and ignition has not been the prime criteria. This paper deals with current work on very high



Fig. 1. Fuel injected at 70 MPa ( $\sim$ 300 m/s) into a supersonic air stream at M = 1.9.

velocity jets with the aim of establishing a fundamental understanding of the processes involved in the break-up, atomization and mixing preparatory to combustion. Much of the work to date has been towards developing a theoretical understanding of the driving processes and an experimental evaluation of the jet, shock interactions. Also, there has been considerable computational modelling of the spray, this being important for understanding some of the details that are very difficult to measure experimentally and to develop submodels for the various codes that have been of enormous importance in modern engine development.

### 2. Diesel fuel jets

An intermittent, diesel fuel jet requires a high pressure upstream of the nozzle that is applied rapidly for a short time period. This can be obtained by two different methods. Either a piston type plunger accelerated rapidly into the fluid filled cylinder or a rapidly opened valve from a more or less constant, high pressure accumulator (common rail) can be used. The former is typical of the mechanical systems in which the plunger is driven by a cam while the latter is representative of the newer common rail types where a solenoid triggered hydraulic valve is used to actuate the start and end of injection. The HEUI types are similar to the common rail types in the supply of the fluid to the driving side of their stepped internal (i.e. within the injector) piston but somewhat like the mechanical systems on its injection side. All injectors have a needle that closes off the nozzle holes at lower pressures to ensure that no unwanted flow occurs. The needle is lifted by hydraulic pressure as the injection pressure is applied. The opening (and closing) pressure is well below the maximum pressure during injection.

Regardless of the system used to obtain the high pressures, the nozzle sac (and any connecting lines to and from the high pressure source) must undergo a re-reflecting wave motion upstream of the nozzles during injection. This will build up the pressure at the nozzle entrance in a series of discrete jumps. For low injection pressures, less than about 100 MPa, this is not particularly significant but it becomes quite pronounced as the injection pressure is raised. The fuel line distortion due to this pulsation is the major reason why pressures are limited in systems with fuel pumps remote from injectors.

The formation of the jet outside the nozzle is therefore subjected to this intermittent pressure rise. It is also a function of the interior design of the nozzle. For a single-hole nozzle with a rounded entry, the flow from sac to spray can be fairly smooth. Real diesel injectors, however, have convoluted internal passageways and sharp edged nozzle entries that have significant effects on the spray. Originally, the spray exiting the nozzle was described as a coherent liquid core for a long period of its development, narrowing as atomization affected its periphery. The core end-location called the "breakup length" was often inferred as being up to 100 times the nozzle diameter. It was assumed that this core gradually decreased in diameter with ligaments, sheets, clumps and droplets being sheared off aerodynamically. Recent papers have indicated that the breakup length is actually quite short and a fully atomized spray exists much closer to the nozzle exit. Several empirical formulae exist for the breakup length, the best known relationships being those of Reitz and Bracco (1982) and Hiroyasu (1995).

After breakup, the spray diverges as a cone of atomized droplets. In the early part of this main spray formation, the physics is somewhat obscure as there may be ligaments, clumps and droplets that continually separate and undergo secondary atomization, collision and merger. Overall, the separation dominates as the diameter grows and the spray reaches a more dispersed state where individual packages of fluid or droplets can be tracked. Droplets are not uniform across the jet, being smaller towards the outer edge. The surface area of the liquid now grows rapidly and it is here that the evaporation and mixing rates become high. Air is entrained into the spray. As the fuel vapour diffuses outwards from the drops and the air inwards, the mixture will eventually fall locally within the flammability range. As long as the temperature and pressure are suitable, as happens due to the high compression ratio in a diesel engine, ignition will then occur.

#### 3. Modelling of diesel sprays

Modelling of fuel preparation is an important component of all internal combustion engine simulation. For thermodynamic models of spark ignition (SI) engines, the fuel/air mixture is usually assumed to be homogeneous, or at least stratified in a known way, and the heat input can be handled from a propagating flame front with calculable characteristics. In diesel (compression ignition, CI) engines, a zonal approach that assumes a heat input related to a fuel injection rate can be used. In neither of these cases are the details of the spray itself critical. However, the spray does have an important effect on the mixture preparation within the cylinder and its details need to be determined during transients in homogeneously charged, SI engines (i.e. the conventional single and multi point injection types), but more particularly in the new gasoline direct injection (GDI) engines and in all diesel engines which are highly stratified. To understand the spray processes, spray modelling needs to be carried out. Further, for 3D, fluid mechanical engine models, a known size, velocity, location and trajectory of the fuel droplets is essential so that the combustion can be located within the appropriate computational cells. Hence, accurate injection/spray models are essential.

Early empirical models were relatively simple. With these, the flow exiting the nozzle is calculated from the known time dependent, injection sac (or line) pressure trace. A quasi-steady modelling using the geometrical nozzle area and a coefficient of discharge then allows the average velocity leaving the nozzle to be determined. Empirical formulas exist for the spray angle, droplet size (Sauter Mean Diameter, SMD) and the related time dependent droplet penetration and velocity (Hiroyasu, 1995; Nishida et al., 1997). The droplet velocity distribution at each angle up to the outer cone may also be evaluated from experimental data. Air entrainment and diffusion models calculating mixture formation are applied until combustion conditions are reached. Models such as this have been used in a number of codes in the past and are still useful either when a simplified solution is required or when the spray contributes only a small quantity of the overall energy. Examples of the latter are those in the dual-fuel combustion simulations for compression ignition engines developed and used at UNSW over a number of years (Choi and Milton, 1997; Mbarawa et al., 2001; Miao and Milton, 2002).

Model development has progressed to a more theoretically based approach. Initial droplet size is a function of the nozzle geometry, diameter and flow with atomization calculated from the aerodynamics of the shear layer between the liquid jet and air. Such aerodynamic breakup follows the classical approach of Weber and Ohnesage and has been well described by Reitz and Bracco (1982). It relies on the generation of Kelvin-Helmholtz instabilities on the jet surface from which droplets emanate. Various formulations such as the WAVE model (Reitz and Bracco, 1982; Reitz and Diwaker, 1996) and the TAB and ETAB (Taylor Analogy Breakup and Enhanced Taylor Analogy Breakup) models, (O'Rourke and Amsden, 1987; Tanner and Weisser, 1998) exist. Droplets are then tracked as the spray develops. More recently, there has been a progression towards inclusion of the flow within the nozzle itself. Turbulence and cavitation initiate and promote the atomization which is then followed by the external aerodynamic effects (Huh and Gosman, 1991; Chaves et al., 1995; Soteriou et al., 1995; Hiroyasu, 1995). This is depicted in Fig. 2. As the sac pressures and hence the velocities increase, these phenomena become more important. Turbulence induced break-up was first modelled by Huh and Gosman (1991), with average turbulence properties giving the mean droplet size and production rate. It has been modified and extended, an example being the augmentation by additional turbulent kinetic energy from the collapse of cavitation bubbles.

The flow into the nozzle sac and then to the nozzle itself is complex. With a sharp edge nozzle, the flow sep-



Fig. 2. Flow in a multi-hole diesel nozzle showing possible jet breakup mechanisms.

arates from the leading edge as it enters the parallel section. At high velocities, this induces a cavitation bubble which grows as injection pressure is increased increasing the atomization and bushiness of the spray. However, there can be detrimental effects. If the separation over-grows and exceeds the length of the nozzle, the flow separates from the nozzle wall throughout its length causing "hydraulic flip", a reversion to a narrow, more coherent spray (Chaves et al., 1995; Soteriou et al., 1995) and the ensuing jet is no longer well atomised. As pressure is further increased, atomization can again improve, probably due to increasing turbulence and aerodynamic interaction. The latest modelling includes the effect of nozzle flow on the emerging jet.

Such an injection simulation is described by Arcoumanis and Gavaises (1998). Here, a one-dimensional, time dependent calculation is used to describe the flow from the fuel pump, through the fuel lines to the injector. The example given is for a mechanical distributor pump application. A CFD model is used to simulate the flow past the needle within the nozzle sac, and then into the parallel nozzle. Exit velocities, based on an effective nozzle area are calculated. The spray model then sets the initial droplet size equal to the effective diameter. Selection of the appropriate atomization model is more difficult. In the work of Arcoumanis and Gavaises (1998), three different models, these being cavitation, turbulence and aerodynamic atomization, were compared to experiment for velocity and SMD. The cavitation model gave the best results these exhibiting the same trends and being generally within 10% of the measured values at two pump speeds. The turbulence model gave similar results for velocity but roughly double for the droplet sizes. The aerodynamic model gave good velocity results in two of the six cases (two pump speeds measured at three axial positions) apparently randomly, while other results were up to 150% of the measurement values. For SMD, the aerodynamic model oversized the droplets by a factor of three. The conclusion in this work is that the cavitation model is the best approach. However, it should not be concluded that the other effects do not contribute.

There are a great many papers on the modelling of high pressure, high velocity subsonic jets and sprays but there is scope for much further work. However, very little modelling has been attempted for supersonic liquid jets, the study by Bloor (1978) being one of the few of significance. For low supersonic jets, his technique incorporated the numerical equations for supersonic flow into the computational solver using a layered, numerical domain. However, it is not clear how far this technique is valid into the supersonic range and many of the details of the jet breakup are unavailable. Recently, as part of the current program, a modelling approach using the FLUENT code was reported by Zakrzewski et al. (2004). Further studies are required both to examine the shock wave formation and its interaction with the jet and to determine whether the empirical formulas developed for high subsonic jets have relevance in the supersonic range.

#### 4. Supersonic and subsonic liquid jets

In any intermittent jet, air is set in motion by the jet head as it emerges from the nozzle and grows in length and diameter. That is, there is a pressure increase ahead of the jet that results in a wave motion being transmitted through the air ahead of it. The highest pressure will exist at the stagnation point at the jet head. The forward air motion from the wave at the side of the jet will be increased by the shear layer and mass transfer from the liquid in the mixing region.

When a subsonic jet injected into quiescent air is compared with a low range supersonic jet (at about 600 m/s, M = 1.8 at ambient conditions), both exhibit a bulbous jet head. In the former, the pressure wave ahead of the jet is not of great significance. Once the jet exceeds sonic velocity, this is no longer the case. Shock waves around the jet become a noticeable feature of the flow. Some of the questions that could be posed are whether the features of the flow (e.g. the breakup length, atomization and penetration distance) follow a similar pattern to subsonic allowing the same empirical equations to be used and whether the shock wave structures of supersonic jets can modify the fuel ignition qualities. Modelling of supersonic jets therefore needs consideration.

#### 5. Experiments on supersonic jets

#### 5.1. Background literature

Supersonic liquid jets have primarily been studied as water rather than fuel jets, the prime purpose being cutting or cleaning. As such, most are continuous rather than intermittent jets. Intermittent jets may exceed continuous jet velocities but are harder to generate. In 1958, Bowden and Brunton (1958) presented a new technique for this purpose. A high-speed projectile was fired into the rear surface of a liquid in a nozzle sac to accelerate it through a nozzle at the sac front to a high velocity, 1050 m/s being reached in their experiments. This technique is now termed the "Bowden-Brunton" method. O'Keefe et al. (1967) using the impact of a 1.77 km/s projectile, measured a water jet velocity of 4.58 km/s. Ryhming (1973), described the process using a onedimensional, incompressible flow analysis. For the analysis of water cannons, Glenn (1975), extended Ryhming's work by including the effect of liquid compressibility. As in O'Keefe's study, the liquid shock wave reflection processes within the nozzle were not considered. In their early work, Field and Lesser (1977), experimenting with oil jets, suggested that a spontaneous combustion of the oil might have occurred at high supersonic speeds. In the light of this, Shi (1994), Shi and Takayama (1999), who were basically examining water jets, included a study of supersonic diesel jets in their experiments. They used a powder gun arrangement similar to the original Bowden-Brunton technique but reported much greater water jet velocities of up to 4 km/s. For diesel fuel jets of more than 2 km/s, they found that smoke covered the test chamber at the completion of a run and postulated that it may have been due to combustion. Holographic interferometry was used to visualize the jets and, while not totally clear, there was some suggestion that additional illumination occurred from the jet.

In a review of his work, Lesser (1995) presented the basic mechanics of supersonic jet generation by using a theory called guided acoustic waves. While it was realized that during the supersonic liquid jet generation process either a single, strong shock wave or multiple shock wave reflections must be involved inside the nozzle cavity, this was ignored in the analysis. In the present work, these effects have recently been included in the analysis and are described by Pianthong et al. (2003).

# 5.2. Apparatus for creating the intermittent supersonic liquid jets

The pressure required to produce a jet is theoretically proportional to the square of the jet velocity but in actuality to a higher index due to increasing losses. Thus, supersonic jets require very high driving pressures, extreme in the high supersonic range. In the low supersonic range, pressures can be obtained by use of the conventional diesel injection systems but high Mach number jets require a different approach. For such jets, rapid pulsing for hours on end as required in an engine presents technical difficulties. However, in the research presented here, the object is to study the fundamentals of supersonic jets and hence a single pulse, short duration jet has been used in the experiments. The same method has been adhered to for consistency throughout the work to encompass low to high supersonic Mach numbers. It is the Bowden–Brunton projectile impact onto the fluid in the nozzle sac.

Collaborative experiments in the program have been carried out at the University of New South Wales, Australia (UNSW) and at the Interdisciplinary Shock Wave Research Center, Tohoku University, Japan (ISWRC). In the UNSW program, the impacting projectile was driven by a vertical, downward firing, single-stage powder gun while the ISWRC apparatus used a larger diameter, two-stage light gas gun. Otherwise, the equipment was substantially the same. The powder gun, Fig. 3, described previously (Pianthong et al., 2002) achieves velocities of up to 1100 m/s for the 8.0 mm diameter, 10 mm long, 0.65 g cylindrical polycarbonate (PC) projectile. The projectile travels downwards through the pressure relief section, which is designed to diminish the blast wave in front of the projectile and to guide it onto the liquid. The nozzle sac is directly connected to it, being seated in the top wall of the test chamber. Mild steel sac/nozzles were used, in a few cases being case hardened. The nozzle exit diameters, d were varied for different experiments from 0.5 mm to 6 mm diameter. As reported (Pianthong et al., 2002), most experiments were carried out with 0.5, 0.7 and 1 mm diameter nozzles with L/d of 2–6. The ISWRC apparatus is similar in concept but different in detail, the vertical two-stage light gas gun firing a larger PC projectile of 15 mm diameter, 20 mm length and weight 4.45 g. A projectile impact velocity of 700 m/s was used, this being the maximum available in the experiments to date although it is expected that these will noticeably increase with optimization.

In both cases, the liquid was retained in the nozzle using a thin plastic diaphragm seal at the top and bottom of the nozzle. Its low strength relative to the impact force of the projectile means that the opening pressure for the nozzle (equivalent to needle lift) is low. Projectile and liquid jet velocities were measured in both systems using a laser beam interruption method, two closely spaced laser beams being placed as close as possible to the nozzle exit in the UNSW experiments. For jet penetration and velocity attenuation measurement, the ISWRC apparatus employed six laser beams.

Nozzle sacs were designed to accommodate the projectile impact. A cylindrical sac with a tapered conical contraction to the cylindrical nozzle was used. While a range of cones were tested, the most common had a 20° included angle (UNSW) and 47° (ISWRC). In the UNSW apparatus, exponential and hyperbolic nozzle convergence profiles were also tested. Additionally, the ISWRC experiments carried out experiments on a stepped (i.e. no conical convergence) orifice to stimulate turbulence and cavitation.



Fig. 3. Apparatus: (a) single-stage powder gun (UNSW) and (b) nozzle details.

#### 5.3. Jet and shock wave shapes from experiment

Shock wave shapes from the conical nozzles at UNSW are shown in Figs. 4 and 5 for a low (600 m/s, M = 1.8) and a high Mach number (1800 m/s, M = 5.3) jet, respectively, both into atmospheric air. At the low jet velocity, the jet emerged with a flat front that changed rapidly with jet growth to the bulbous profile similar to that found in subsonic jets. A detached shock wave was clearly visible ahead of the jet from the time of its formation. At higher Mach numbers, the shock wave became attached to the jet head forming an oblique system similar to solid body interactions. This distorted the jet head, making it sharper as the jet velocity rose as can be seen in Fig. 5. Some random irregularities, evident in

the photographs, occurred in the jet head that may have been due to turbulence generated within the nozzle sac or to distortion of the nozzle itself caused by the high pressure. However, as will be discussed later, the pressure maximized after the initial jet was formed and so nozzle distortion is more likely to affect the later portions of the jet rather than the jet tip. Hence, the fluid mechanics inside the nozzle is the most likely cause. Also of note was that, at the higher Mach numbers, a secondary shock wave existed alongside the jet, as indicated by the arrows in the centre photograph.

Only high Mach number jets (1800 m/s, M = 5.3) were examined at ISWRC, these being depicted in Fig. 6. Comparing these profiles with those of Fig. 7(a) for hardened nozzles suggests that the scaling of the



Fig. 4. Water jet from a conical 5 mm. Nozzle at M = 1.8 (600 m/s) showing the jet development and detached bow shock wave (UNSW experiments).



Fig. 5. Diesel fuel jet from a 1 mm. Nozzle at M = 5.3 (1800 m/s) showing the jet development (UNSW experiments).



Fig. 6. Diesel fuel jet at M = 5.3 (1800 m/s) showing the jet development (ISWRC experiments).



Fig. 7. Diesel fuel jets from: (a) hardened, conical nozzle, 2350 m/s; (b) hyperbolic nozzle, 1850 m/s and (c) stepped nozzle, 1560 m/s showing the jet development. Experiments (a) and (b) are UNSW; (c) is ISWRC. All nozzles 1 mm diameter.

experiment has only a small effect. The difference between Figs. 5 and 6 profiles is probably due to the stainless steel nozzles used at ISWRC. In the ISWRC experiments, the secondary shock wave system is clearer and a tertiary system is also present. The latter are also visible in some of the UNSW experiments. Even at high Mach numbers with an attached bow shock, the head still tended to form a bulbous shape, narrowing behind to the long core. It then thickened again at the secondary shock positions. In all cases, near the nozzle exit, a spray zone existed with a higher conical included angle than the flow ahead of it. Note that it is possible that hydraulic flip has occurred in some cases and this needs further investigation.

The effect of nozzle differences is illustrated in Fig. 7. This shows jets from a hardened conical nozzle, a mild steel hyperbolic nozzle and a stepped nozzle. The hardened nozzle exaggerated the patterns discussed above with the long, narrow core clearly visible. The hyperbolic (and exponential) nozzles showed a smoother, more uniform jet shape indicating that random effects from the flow into the sharper, conical entry carry over into the spray. The jet head from the stepped nozzle was the most irregular and bulbous. Even at the highest initial velocities of around 1560 m/s, the flatter head reduces the conical shock to a more rounded shape. At 126 µs, it is attached whereas at 226 µs, it has separated and thereafter moves progressively ahead of the jet. This is likely to be due to both the broader spray head and the more rapid attenuation after the first 100-200 µs.

## 5.4. Jet velocities

One of the immediate aims of the experiments was to obtain the maximum jet velocity. As noted above, jet velocities of up to 4000 m/s had been reported from similar techniques. The present experiments both at UNSW and ISWRC were unable to achieve such values. At the maximum current capability, as the nozzle size was decreased, the jet velocity rose. However, the maximum that could be obtained in these experiments were as in Table 1.

A nozzle of area ratio 130:1 gave the best results. A smother nozzle shape provided small increases while a stepped nozzle reduced the velocity. The nozzle material was of greater significance, the hardened nozzle showing significant improvement. However, these nozzles cracked during the run and were not reusable. The normal, mild steel nozzles exhibited some distortion but were not destroyed in a single test. Measurements indicated that reuse was possible although in the interest of accuracy, this was not done. For the low Mach number (M = 1.8) tests, nozzles showed no permanent distortion or erosion. Hence, they could be used for many runs.

Table 1 Maximum measured jet velocities (to nearest 50 m/s)

#### 5.5. Jet attenuation and penetration

Using the series of six lasers (and a high speed camera, results not shown here) in separate tests at ISWRC, the profile of jet penetration with time was determined. Differentiating this gives the jet velocity/time relationship from which the attenuation profile was plotted as in Fig. 8(a). There is a suggestion that in fact the velocity oscillates about the smoothed curve depicted here due to the pulsations within the nozzle sac and this is being



Fig. 8. Time dependent jet velocities and jet penetration for supersonic jets: (a) experimental measurements of diesel and water jets from conical nozzles; (b) calculated values from conventional subsonic formulae given by Nishida et al. (1997).

Nozzle profile	Nozzle area ratio (dia. mm)	Nozzle material	Tests at	Max.velocity, m/s
Conical	256:1 (0.5)	Mild steel	UNSW	1850
Conical	130:1 (0.7)	Mild steel	UNSW	2000
Conical	64:1 (1)	Mild steel	UNSW	1800
Conical	130:1 (0.7)	Hardened	UNSW	2500
Conical	225:1 (1)	Stainless steel	ISWRC	1800
Hyperbolic	64:1 (1)	Mild steel	UNSW	1900
Exponential	64:1 (1)	Mild steel	UNSW	1850
Stepped	225:1 (1)	Stainless steel	ISWRC	1560

further explored by use of a high-speed camera in separate experiments.

The initial velocity of the diesel fuel jet was slightly higher than that of water, the exact values being 1863 m/s and 1714 m/s respectively. The diesel velocity attenuation was slightly lower, particularly after the first 300 µs although by 2000 µs there is little difference. Hence the diesel penetrates further for most of the injection period. The reduction in velocity is quite high during the first 300 µs. A comparison (Pianthong et al., 2003) of these test results with the estimation of jet velocity attenuation and penetration distance obtained from the use of a conventional, empirical diesel fuel spray formula of Nishida et al. (1997) is shown in Fig. 8(b), calculated for an initial diesel velocity of 1800 m/s. The trends are similar although the experiments indicate higher values of both velocity attenuation and penetration by 20-30%. This suggests that the subsonic formula, while needing some improvement, provides a reasonable starting point for estimates. Some penetration estimates at 1 ms (a typical ignition delay time in a diesel engine) for 600, 1200 and 1800 m/s jets are about 117, 161 and 194 mm respectively. These are moderately large distances and may place a limitation on the use of extreme pressure injection in smaller engines depending upon whether the shock waves can further shorten the ignition delay. Note that the stepped nozzle has a lower initial velocity and higher attenuation and therefore does not penetrate so far.

#### 5.6. Autoignition of the fuel

As previously noted, there have been several suggestions (Field and Lesser, 1977; Shi, 1994; Shi and Takayama, 1999) that autoignition of diesel fuel and other oils is possible at ambient air temperatures in a supersonic jet with velocity around 2000 m/s. This is due to the shock wave heating of the air in the mixing zone. If correct, it could have implications for improved ignition and combustion in marginal conditions and for the use of low cetane number fuels in compression ignition engines.

This concept has arisen due to the appearance of smoke in the test chamber after supersonic injection of such fuels. Also, an analysis based on the visualized shock and jet shapes and a solid-body analogy indicates that extreme conditions are possible. This is shown in Fig. 9. Here, if a blunt body, normal shock exists on the jet leading edge, the conditions are well above those required for autoignition. If the shock is oblique and attached, the conditions at the side of the jet are much less severe. While these are elevated, and hence would enhance ignition characteristics, they would not be sufficient to provide ignition without additional heating of the air.

An extensive experimental examination was carried out in the UNSW facility to determine if autoignition of the fuel occurred at lower than normal diesel engine ignition conditions. Tests covered a wide range of conditions, jet velocity ranging from 1800 to 2000 m/s, test chamber air heated from ambient to 100 °C and fuels from conventional diesel with a cetane number, CN of 45 to pure cetane (CN = 100). Note that, as cetane number increases, so does the ease with which the fuel autoignites. Measurement techniques looked for visible light from a combustion source by exposure to photographic plate with the test chamber darkened and sampling of the post combustion gases for traces of carbon dioxide (CO<sub>2</sub>), carbon monoxide (CO) and nitrogen oxides,  $(NO_{y})$ . Overall, it was assessed that, for these jets, no autoignition resulted. There was ample evidence of post-test smoke in the test chamber but this was due



Fig. 9. Assessment of jet and air conditions in the mixing zone using a solid body supersonic flow analogy.

to very high levels of fuel vaporisation. The tests do not as yet indicate that low temperature autoignition is not possible from the shock wave heating. A different nozzle geometry or higher jet velocities may contribute. However, the additional evaporation does indicate that the fuel may be better prepared for ignition than with a conventional spray. Currently, tests are underway to examine the ignition enhancement of low cetane number fuels (i.e. fuels not suitable for direct use in diesel engines) such as butane, propane, ethane and eventually methane to see if supersonic jets can improve their ignition characteristics.

#### 6. Numerical assessment of driving supersonic jets

#### 6.1. One-dimensional theory

A general method of predicting the driving pressure from the projectile impact and relating it to the jet velocity is a valuable tool and can help to explain such phenomena as multiple shock wave formation. To do this, a one-dimensional model of the shock wave pressure generation within the cavity has been developed (Pianthong et al., 2003a). It requires initial input conditions of projectile mass and velocity at the time of impact, and the liquid mass. The nozzle area was assumed to consist of a single step from that of the sac. On impact, a shock wave moving in the projectile direction is generated in the liquid while another in the projectile is transmitted in the opposite direction. By equating the momentum transfers and the interface velocity, equations can be developed which give the pressure, P, behind the liquid shock and hence the related shock velocity.

Normal reflection of this shock from the end-wall of the cavity determines the first pressure pulse that drives the nozzle flow. A series of reflections between the end wall and projectile face incrementally increase the driving pressure at the nozzle entrance. At each reflection cycle, the projectile velocity is assumed to be that of the previous interface. That is, the projectile, liquid interface remains coherent throughout. Each pulse slows the projectile incrementally.

As the projectile and liquid have the same cross-sectional area, the length ratio of the liquid in the nozzle sac to that of the projectile is important. Depending on this value, different wave combinations are possible. This is because there is a maximum quantity of liquid that can be accelerated to conditions compatible to the liquid shock by the original momentum decrement of the projectile. For example, if the liquid slug is very long, the projectile must slow further after it has transferred the appropriate momentum quantity to a value below that of the particle velocity behind the shock. If it is very short, the reflected shock from the end of the nozzle sac will re-reflect from the projectile interface while it is still transferring the original momentum quanta. In between, a situation will exist where the returning (reflected) shock in the liquid just reaches the projectile as the last of that momentum increment is being transferred. This is referred to here as the "balanced" length, depicted in Fig. 10, and used in these calculations. Other combinations will give slightly different answers but require a more complex formulation. For the balanced case, a proportion, F, of the total momentum decrement is transferred to the liquid during the forward motion of the shock, the remainder during its return to the projectile, the total time being  $t_c$ . This time,  $t_c$  is also that for the shock in the solid to traverse its length. The ratio of the time period for the forward motion of the shock in the liquid until the impact location,  $t_f$ , to that for the shock in the solid to traverse its length,  $t_c$ , determines F.

Assumptions are that normal impact and one-dimensional particle and shock motion occur, that the interface between projectile and liquid in the nozzle is coherent, the nozzle has a single step from sac to exit, and the walls are rigid. The shock wave velocity in both the projectile and liquid, particle velocity, and impact pressure rise P can then be estimated.

$$P = F \rho_s C_s (V_s - U_{p2}) = \rho_1 C_2 U_{p2} \tag{1}$$

Here,  $\rho$  is the density, C the shock velocity. Subscripts (s) relate to the projectile, (1) and (2) to the liquid before



Fig. 10. Balanced case for shock motion in the nozzle sac.

and after impact while  $V_s$  is the initial projectile velocity and  $U_p$  (i.e.  $U_{p2}$ ) the liquid particle velocity after impact. By applying the shock Hugoniot relation, the shock wave velocity in both substances can be obtained as in Eq. (2).

projectile  $C_s = a_s + k_s(V_s - U_{p2})$ liquid  $C_2 = a_2 + kU_{p2}$  (2)

Here *a* is the sound velocity in the substance and *k* (i.e. *k* for the liquid,  $k_s$  for the projectile) is a material constant. A solution for  $U_{p2}$ , can then be found from which the impact pressure P follows. On reflection of an incident shock wave, a normal reflected shock wave of velocity  $C_3$  returns towards the projectile. Determination of the driving pressure  $P_z$  and particle velocity  $V_z$  just upstream of the nozzle exit follows using conventional, normal shock wave reflection techniques, as described in Pianthong et al. (2003a). The jet velocity,  $V_i$  is then obtained by integrating the one-dimensional Euler momentum equation between the nozzle and atmospheric conditions using the Tait equation of state relationship. Subscripts 'z' and 'a' designate the nozzle and atmospheric conditions respectively while  $A_w$  and n are constants for water (values being 363.2e + 6 and 6.11, respectively).

$$V_{j}^{2} = V_{z}^{2} + \frac{2n}{n-1} \frac{(P_{a} + A_{w})^{\frac{1}{n}}}{\rho_{a}} \left\{ (P_{z} + A_{w})^{\frac{n-1}{n}} - (P_{a} + A_{w})^{\frac{n-1}{n}} \right\}$$
(3)

Basically, the numerical work required is simple, involving the simultaneous solution of 12 algebraic equations. These relationships describe the complete cycle from the onset of the impact until the shock wave reflected from the end-wall reaches the projectile at time  $t_c$ , reducing its velocity. This transfers further quantum of momentum to the liquid. Reflection cycles are then repeated until the projectile eventually stops. The x-t diagram of Fig. 11 describes the shock reflection in the nozzle sac. At this stage, the liquid has been taken as water and comparisons made with water jet experiments. This is because the very high pressure properties into the range considered here of water are known whilst those of diesel fuel, a complex compound, are not. A comparison of measured and calculated jet velocities assuming a coefficient of velocity,  $C_v$  of 1 for the latter is shown in Fig. 12. A typical value of  $C_v$  of 0.63–0.72 is then obtained for use with the calculations by matching values to the experiments. This is very reasonable for a pulsed nozzle.

#### 6.2. Estimation of jet velocity for different conditions

Using a similar calculation procedure, projectile characteristics can now be considered (Milton and Pianthong, 2004). Nozzle velocity coefficients are assumed to remain unaltered while the calculations are for a 1 mm orifice (area ratio 64:1).

Fig. 11. Typical *x*-*t* diagram for the nozzle sac, liquid shock wave reflections projectile velocity 1100 m/s.

13

U<sub>p13</sub>

10

U<sub>p10</sub>

7

U\_\_\_\_

4 U

C<sub>13</sub>

C<sub>10</sub>

C-7

C<sub>4</sub>

1

transmitted shock wave

in nozzle orifice

12

9 C<sub>9</sub>

6

3

1

C<sub>12</sub>

11

C₁

8

C,

5

reflected shock wave

in nozzle cavity

С

С

C<sub>6</sub>

projectile or particle velocity

U<sub>p</sub>

2800

2



Fig. 12. Measured and calculated jet velocities for a range of nozzle diameters.

Interestingly, the projectile size does not alter the result as long as it remains of the same material. Hence, the basic parameter affecting the jet velocity is the projectile impact velocity. This is shown in Fig. 13. To obtain a jet velocity of 3000 m/s on the fourth pulse, assuming a realistic  $C_v = 0.63$ , requires an impact velocity of 2500 m/s. For a 4000 m/s jet, the projectile would have to reach a velocity of at least 3800 m/s. This is extremely high.

The nozzle sac pressures at these velocities become immense. These are shown in Fig. 14. For a 3000 m/s

 $V_{j4}$ 

V<sub>j3</sub>

V<sub>j2</sub>

 $V_{j1}$ 



Fig. 13. Predicted relationship between projectile and jet velocities.



Fig. 14. Estimated nozzle sac pressures to obtain extreme jet velocities.

jet, the values reached would be 18.5 GPa (pulse 4) while for 4000 m/s it would be about 34 GPa. The latter is impossibly high. The effects on the liquid at these pressures are unknown even for water but it is unlikely that the Tait equation would still represent its properties while the effect on the projectile is difficult to estimate. Even at the lesser 3000 m/s jet velocity, it is likely that the nozzle would distort or shatter well before the peak jet velocity was reached. Also, the pressure wave system within the walls of the nozzle would become significant. These effects would further reduce the coefficient of velocity and the potential of the system to generate high jet velocities.

#### 6.3. Computations from the Autodyne code

The two-dimensional interactive non-linear dynamic analysis software (AUTODYN-2D TM, Century Dynamics Inc.) was used to simulate this driving event (Pianthong et al., 2003b). This numerical code can treat the Lagrangian and Eulerian frames in a fully coupled way providing a great flexibility in simulating complex wave and material interactions among different phases. The dimensions and the projectile speed used in the simulation of 1100 m/s matched the UNSW facility. The hydrodynamic behavior of the different materials in different coordinate systems is solved in a fully coupled way using the code. The numerical code has a data library and detailed mechanical properties of most typical materials such as those used in the current study are down loaded automatically during the calculations.

The code has been used to calculate the specific case of a 1100 m/s projectile as in the UNSW experiments. The calculation is taken over a time ranging from impact to 25 ms, well after the formation of the jet. The code clearly shows the shock formation in the liquid, where the pressure and velocity in the sac are 1 GPa and 500 m/s respectively. These conditions are close to the one-dimensional analysis predictions. When the shock reflects from the nozzle end, the jet formation commences. Repeated reflections raise the sac pressure and increase the jet velocity, the latter reaching to a maximum of 1420 m/s at about 14 µs. Note that this velocity is less than measured in the experiments while the one-dimensional analysis is, more realistically, greater. This is most likely to be due to the code predicting a projectile bounce at about the time maximum velocity is obtained. Experiments do not confirm this as the projectile was usually found embedded firmly into the nozzle sac at the end of the test. This code shows the wave effects in the nozzle material which is important as distortion may modify the velocity and alter the spray characteristics. The use of the code is beneficial although further exploration is necessary.

#### 7. CFD examination of the supersonic liquid spray

Previously, in the simulation programs used for diesel and dual fuel combustion at UNSW, empirical or quasiempirical models of the diesel injection were used. For the present research, it was felt that these were not suitable for the following reasons. First, the shape of the conical bow shock wave around the jet was needed to compare with shadowgraph experiments, the conditions generated by it being important for a further understanding of the fuel/air mixing preparatory to combustion. Second, the shape of the jet itself was critical in validating the CFD approach from the experimental results. Finally, as many factors related to intermittent, supersonic jets such as the breakup and atomization processes were unknown, validation would best be obtained by comparisons with steady, supersonic jet cases. A conventional CFD approach could more easily work through such a sequence. Hence, the proprietary CFD code, FLUENT was chosen. Experimental studies are unable to capture intricacies within the jet core close to the nozzle and are intrusive leaving details of the internal structure of the jet open to question. The use of CFD has been advantageous in starting to evaluate these. However, even CFD presents many difficulties.

This aspect of the research has been recently reported (Zakrzewski et al., 2004) in this journal and hence will only be summarised here. Obtaining converged solutions from the CFD code for this complex phenomenon proved to be extremely difficult and the following procedure was used to solve these problems and to progressively validate a solution. In modelling such a complex process as a transient, supersonic deformable surface, factors considered were the shock wave formation ahead of the jet and the density variations from the surrounding air, through the mixing layer to the liquid core. While several turbulence approaches were examined, the  $k-\varepsilon$  model was found to be the only suitable solution due to convergence problems with the others.

The approach followed has been:

- To use a computational domain from the nozzle exit for a wide field around the jet.
- To assume that the inflow conditions are steady for the lifetime of the jet.
- To develop the solution procedure in order against
  - steady, supersonic flow of air over solid bodies of jet-like shapes;
  - steady, supersonic flow of air over water vapour jets;
  - steady, supersonic flow of air over liquid jets;
  - the flow of unsteady, supersonic vapour jets into air;
  - the flow of unsteady, supersonic liquid jets into air.

The steady-state, solid body solutions obtained by Zakrzewski et al. (2002), used an arbitrarily defined liquid boundary onto which the air flow impinged. This procedure provided the easiest validation as the shock wave system is well defined experimentally from the shadowgraph experiments. Typical results are shown in Fig. 15. Note that the higher Mach number solution provided the best agreement, probably due to a better choice of jet profile, these being themselves selected from experimental photographs. The jet core seems to be more coherent at the high Mach number of 5.9, possibly due to the jet being "flipped" at this value. Hence, defining a boundary in the M = 1.8 case presented greater difficulties due to the well atomized layer around the head of the liquid core.

Unsteady vapour jet solutions provided good insight into both the physics of the modelling, the jet shapes to be expected and the numerical procedures to be followed. This is fully explained in Zakrzewski et al. (2004). A progression was then made to the unsteady, liquid jet solution which required considerable computational effort and to date, only a single case has been run. This was for the 600 m/s jet (M = 1.8), computational times for higher velocities being extreme. The solution obtained at this velocity compared well with the experiments although the surface profile was much less disturbed by aerodynamically generated wave fronts. The comparison can be seen in Fig. 16.

The modelling approach with both the one-dimensional time dependent internal flows and the AUTO-DYNE and FLUENT codes has highlighted some of the physics involved in the jet driving processes. The fluctuating pressures throughout the jet lifetime that explain the pulsations observed in practice, are now evident and profiles of the two-phase, air-fluid regions in the mixing region are available. Typical profiles at different stages of the jet development for the latter were given in Zakrzewski et al. (2004). While these still require improved and extended computations and further validation, they provide a starting point for the evaluation of the atomization and mixing of supersonic jets.



Fig. 15. Comparison of experiment and solid body, supersonic jet computations using FLUENT for: (a) M = 1.8 jet and (b) M = 5.9 jet.



Fig. 16. Comparison of simulated and experimental jet shapes for the unsteady liquid jet at M = 1.8 (600 m/s).

#### 8. Conclusions

This paper has summarised work on pulsed, supersonic liquid fuel jets. These have been studied experimentally for low and high range supersonic Mach numbers and simulated for the low Mach numbers only. Both experiments and simulation are complex. The experiments have evaluated the velocities obtainable, the shock wave and jet structure in the external flow and the potential for autoignition of diesel type fuels. Nozzle shapes and materials have been evaluated and the jet attenuation and penetration studied. The latter show some similarity to the trends calculated from empirical formulas used for high subsonic diesel jets although the specific values need to be reassessed.

Several new phenomena have been found, these being, in particular a secondary and tertiary shock wave system. The numerical studies have used a one-dimensional analysis and the Autodyne code for assessment of the internal pressure rise due to the impact of the driving projectile. The one-dimensional approach provides answers with values higher than experiment that fit a realistic coefficient of velocity of between 0.6 and 0.7. The Autodyne code gives values slightly lower than experiment in the later stages of the jet but is useful in that it also simulates the shock wave pressure rise through the nozzle walls.

For the external jet, the FLUENT code has been used. Its application has taken considerable development and has progressed through a solid body assessment of the bow shock waves for both low and high supersonic cases, steady vapour and liquid jets and unsteady vapour and liquid jets for the low supersonic case only at this stage. While some differences in the overall shock wave patterns and jet shape exist, particularly in regard to the former, the results are generally in reasonable agreement with experiment. This allows such things as the mixing layer to be evaluated which is important for further incorporation into engine combustion codes. It must be emphasized that, at this stage, results are still preliminary and more development is required.

The value of increasing fuel injection pressures above current values for fuel jets in engines is still unclear. Some experiments indicate that there are further benefits on atomization for pressures above about 250 MPa, others do not. However, in the 250-300 MPa range usually considered, the shock wave effects are either non-existent or negligible. A considerable increase in magnitude of the jet may be significant due to the formation of strong shock waves as shown here. The full effect of these waves remains unclear. While no autoignition was found in the current series of tests, fuel vaporisation was certainly enhanced. Thus, promotion of autoignition is likely and further tests, starting from known combustion conditions (e.g. about 3 MPa and 500 °C ambient) should be considered. Increasing the jet velocity while working downwards in pressure and temperature would define the autoignition envelope for such high pressure jets.

On the fundamental side, good spray atomization is essential. The single-hole, conical or smooth nozzles used in these experiments seemed to promote some additional atomization although a good comparison with subsonic equivalents is needed. The sharp edge nozzle changed the spray pattern significantly, making it bushier. Nozzles with passageways approaching those of diesel injectors are under consideration with studies to maximise the cavitation within the nozzle. Further experiments with these in the low supersonic range are to be carried out.

Improved modelling of the jet development processes is essential. Much has been learnt from the current approach both of the flow within the nozzle sac from the one-dimensional analysis and the Autodyne code. The same is true for the external flow modeled by the Fluent CFD code in relation to the shock structures and a preliminary assessment of the mixing. However, these approaches now need to be coupled so that the inflow boundary to the spray incorporates the unsteady velocity. A fully transparent atomization model needs to be incorporated and the relative effects of aerodynamic shear, turbulence and cavitation evaluated. It may beneficial to develop a numerical scheme specific to this problem.

The use of very high pressure jets as used in jet cutting may have limitations in engines due to the potential for damage, either to the nozzle, cylinder walls or piston. However, jets used for cutting purposes are coherent and the greater spray angle and finer atomization of turbulent or cavitating nozzles should reduce this problem. The measured penetration distances, even with the 2000 m/s jets, are not unreasonable. Jets of around 600–1000 m/s should further minimise potential damage. The maximum injection pressures that can reasonably be used need to be explored. Jet velocity, engine size and air swirl need to be matched in any consideration of this type.

At present, further experiments are being carried out using more volatile fuels, these being butane, propane ethane and methane liquefied at low temperatures. These are all alternative fuels with low cetane numbers. Enhancing their ignitability would be beneficial.

Finally, an experimental and numerical study of supersonic fuel jets injected as a cross-flow into a supersonic airstream needs to be undertaken. Very complex interactions will occur that may enhance scramjet combustion using liquid fuels. Modelling the shock wave interactions alone would provide a considerable and interesting task.

#### Acknowledgements

The authors would like to acknowledge the major contributions of Dr. S. Zakrzewski, Professor M. Behnia, Professor K. Takayama and Professor T. Saito on work directly related to supersonic jets, Professor S. Xu and Professor R.D. Archer for their work on subsonic injection into supersonic airstreams and Dr. I. Choi, Dr. M. Mbarawa and Dr. H. Miao for their earlier development of diesel jet and spray models.

#### References

- Arcoumanis, C., Gavaises, M., 1998. Linking nozzle flow with spray characteristics in a Diesel Fuel Injection System. J. Atomizat Sprays 8, 307–347.
- Bloor, M.I.G., 1978. Hypersonic liquid jets. J. Fluid Mech. 84 (2), 375–384.
- Bowden, F.P., Brunton, J.H., 1958. Damage to solids by liquid impact at supersonic speeds. Nature 181, 873–875.
- Bowden, F.P., Brunton, J.H., 1961. The deformation of solids by liquid impact at supersonic speeds. Proc. R. Soc. London Ser. A 263, 433–450.
- Chaves, H., Knapp, M., Kubitzek, A., Obermeier, F., Schnieder, T., 1995. Experimental study of cavitation in the nozzle hole of diesel injectors using transparent nozzles. SAE Paper 950290.
- Choi, I.S., Milton, B.E., 1997. A dual-fuel combustion model for premixed natural gas with distillate ignition in a quiescent bomb. Numer. Heat Transfer, A 31 (7), 725–743.
- Cummins, C.L., 1993. Diesel's Engine—vol. 1, from Conception to 1918. Carnot Press, Oregon.
- Field, J.E., Lesser, M.B., 1977. On the mechanics of high speed liquid jets. Proc. R. Soc. London Ser. A 357, 143–162.
- Glenn, L.A., 1975. The mechanics of the impulse water cannon. Comput. Fluids 3, 197–215.
- Hiroyasu, H., 1995. Fundamental spray combustion mechanism and structures of fuel sprays in diesel engines. In: Chui, H.H., Chigier,

N., (Eds.), Mechanics and Combustion of Droplets and Sprays. Begell House, NY.

- Huh, K.Y., Gosman, A.D., 1991. A phenomenological model of diesel spray atomization. In: Proceedings of the International Conference on Multiphase Flows, Tsukuba, Japan.
- Jenkins, D.C., 1955. Erosion of surfaces by liquid drops. Nature 176, 303–304.
- Lesser, M., 1995. Thirty years of liquid impact research. Wear 186, 28-34.
- Mbarawa, M., Milton, B.E., Casey, R.T., 2001. Experiments and modelling of natural gas combustion ignited by a pilot diesel fuel spray. Int. J. Therm. Sci. 40, 927–936.
- Miao, H., Milton, B.E., 2002. Modelling of the gas/diesel dual-fuel combustion process for conditions applicable to engines. Numer. Heat Transfer, A 41, 725–739.
- Milton, B.E., Pianthong, K., 2004. Prediction of the driving conditions for hypersonic liquid fuel jets. In: Proceedings of the International Symposium on Shock Waves, ISSW24, Beijing.
- Nishida, K., Ochiai, H., Arai, M., Hiroyasu, H., 1997. Characterization of diesel fuel spray by ultrahigh-pressure injection. Trans. Jpn. Soc. Mech. Eng. B 63 (605), 344–349 (in Japanese).
- O'Keefe, J.D., Wrinkle, W.W., Scully, C.N., 1967. Supersonic liquid jets. Nature 213, 23–25.
- O'Rourke, P.J., Amsden, A.A., 1987. The TAB method for numerical calculation of spray droplet breakup. SAE Paper 872089.
- Pianthong, K., Zakrzewski, S., Milton, B.E., Behnia, M., 2002. Supersonic liquid jets: Their generation and shock wave characteristics. Shock Wave J. 11 (6), 457–466.
- Pianthong, K., Milton, B.E., Behnia, M., 2003a. Generation and shock wave characteristics of unsteady pulsed supersonic liquid jets. J Atomizat. Spray 13 (5&6), 475–498.
- Pianthong, K., Milton, B.E., Behnia, M., 2003b. Fundamentals of supersonic diesel fuel jets for use in diesel engines. In: Proceedings of the 12th International Pacific Conference on Automotive Engineering, Thailand, Paper D08.
- Reitz, R.D., Bracco, F.V., 1982. Mechanism of atomization of a liquid jet. Phys. Fluids 24 (2), 1730–1741.
- Reitz, R.D., Diwaker, R., 1996. Effect of droplet breakup on fuel sprays. SAE paper 960469.
- Ryhming, I.L., 1973. Analysis of unsteady incompressible jet nozzle flow. J. Appl. Maths. Phys. (ZAMP) 24, 149–164.
- Shi, H.H., 1994. Study of hypersonic liquid jet. Ph.D. Thesis. Tohoku University, Japan.
- Shi, H.H., Takayama, K., 1999. Generation of hypersonic liquid fuel jets accompanying self combustion. Shock Wave J. 9, 327–332.
- Soteriou, C.C.E., Andrews, R.J., Smith, M., 1995. Direct injection diesel sprays and their effect of cavitation and hydraulic flip on atomization. SAE paper 950080.
- Tanner, F.X., Weisser, G., 1998. Simulation of liquid jet atomization for fuel sprays by means of a cascade drop breakup model. SAE paper 980808.
- Xu, S., Archer, R.D., Milton, B.E., 1999. Injection of liquid hydrocarbon jets normal to supersonic airflows. In: Proceedings of the International Symposium Shock Waves, ISSW22, London, pp. 1525–1530.
- Zakrzewski, S., Behnia, M., Milton, B.E., 2002. A blunt body analogy for bow shock characteristics in front of a supersonic liquid jet. In: Proceedings of the 2nd International Conference on Computational Fluid Dynamics, ICCFD2, Sydney.
- Zakrzewski, S., Milton, B.E., Pianthong, K., Behnia, M., 2004. Supersonic liquid fuel jets injected into quiescent air. Int. J. Heat Fluid Flow 25 (5), 833–840.