

Fundamental Aspects of Jet Ignition for Natural Gas Engines

Epaminondas Mastorakos, Patton Allison, Andrea Giusti, and Pedro De Oliveira
University of Cambridge

Sotiris Benekos, Yuri Wright, and Christos Frouzakis
ETH Zurich

Konstantinos Boulouchos
Swiss Federal Institute of Technology

ABSTRACT

Large-bore natural gas engines may use pre-chamber ignition. Despite extensive research in engine environments, the exact nature of the jet, as it exits the pre-chamber orifice, is not thoroughly understood and this leads to uncertainty in the design of such systems. In this work, a specially-designed rig comprising a quartz pre-chamber fit with an orifice and a turbulent flowing mixture outside the pre-chamber was used to study the pre-chamber flame, the jet, and the subsequent premixed flame initiation mechanism by OH* and CH* chemiluminescence. Ethylene and methane were used. The experimental results are supplemented by LES and 0D modelling, providing insights into the mass flow rate evolution at the orifice and into the nature of the fluid there. Both LES and experiment suggest that for large orifice diameters, the flow that exits the orifice is composed of a column of hot products surrounded by an annulus of unburnt pre-chamber fluid. At the interface between these layers, a cylindrical reaction zone is formed that propagates in the main chamber in the axial direction assisted by convection in the jet, but with limited propagation in the cross-stream direction. For small orifice diameters, this cylinder is too thin, and the stretch rates are too high, for a vigorous reaction zone to escape the pre-chamber, making the subsequent ignition more difficult. The methane jet flame is much weaker than the one from ethylene, consistent with the lower flame speed of methane that suggests curvature-induced quenching at the nozzle and by turbulent stretch further downstream. The velocity of the jet is too high for the ambient turbulence to influence the jet, although the latter will affect the probability of initiating the main premixed flame. The experimental and modelling results are consistent with ongoing Direct Numerical Simulations at ETH Zurich.

CITATION: Mastorakos, E., Allison, P., Giusti, A., De Oliveira, P. et al., "Fundamental Aspects of Jet Ignition for Natural Gas Engines," *SAE Int. J. Engines* 10(5):2017, doi:10.4271/2017-24-0097.

INTRODUCTION

Flame initiation by a turbulent jet emerging from ignition in a pre-chamber, often denoted as "Turbulent Jet Ignition" (TJI) in the literature, is increasingly being used in large-bore lean natural gas engines. The thorough review by Toulson et al. [1] should be consulted for most of the engine-based findings until 2010. This technology is claimed to have various advantages for high pressure, large-bore systems that are difficult to ignite by conventional electrical sparks due to the lean equivalence ratio of the mixture and the higher energy needed for breakdown. With TJI, the emergence of jets from the pre-chamber may offer multiple ignition sites for the lean premixed flame in the main chamber, which allows for quick burn rates. In addition, misfires may be avoided because the pre-chamber mixture may be made richer than in the main chamber by extra fuel injection and the spark can be protected from violent flow [1].

Most of the research on TJI has been performed in engine environments, where the small scales and high pressures make imaging difficult and hence the details of the process are still not fully understood. The turbulent jet ignition problem has been extensively studied in simpler rigs. Yamaguchi et al. [2] working with a divided chamber bomb and propane mixtures, investigated the ignition and burning mechanisms of the main chamber mixture by a torch jet (orifice diameters between 4 and 14mm). The nozzle diameter and the volume ratio were found to have a strong influence on the structure of the torch jet independently of the main chamber mixture ratio. Recently, Biswas et al. [3] studied the ignition mechanisms of CH₄/air and H₂/air mixtures and, similarly to Yamaguchi et al., two ignition mechanisms were identified that were called "jet ignition" and "flame ignition", corresponding to jets consisting of hot combustion products and wrinkled turbulent flames, respectively. They observed that the ignition took place at the side surface of the hot jet during the deceleration process and that increased orifice diameters or increased pressure led to the "flame ignition" regime.

These authors introduced the idea of a critical global Damköhler number, as a limiting parameter that separates the ignition from a no-ignition regime.

Studies in rapid compression machine (RCM) have also been conducted [4,5,6]. Gentz et al. [4] studied the effect of nozzle diameter as well as the number of orifices. The study of the multi-nozzle configurations revealed that at near stoichiometric conditions nozzles that produce more spatially distributed jets result in faster combustion progression, while at lean conditions, nozzles of smaller diameters that produce more vigorous jets are needed to initiate combustion. Gholamisheeri et al. [5] conducted experiments in a RCM utilizing high speed imaging. Compared to Ref. [4], the authors of this study focused more on the influence of orifice diameter and mixture composition, on the jet inlet, tip speed and the rate of flame spread in the main combustion chamber. The results show that reducing the orifice diameter led to a decrease in the hot jet penetration speed.

Despite these contributions, the details of the flow and flame passage through the orifice, which seems crucial for determining overall ignition and the ignition mechanism, have not been examined extensively yet. In addition, the details of flame inside the pre-chamber have not been explored. The aim of the present work is to better understand the canonical problem of jet ignition through simplified experiments and modelling outside of the complex engine environment, in the hope that the key physical mechanisms revealed will help interpret engine observations and other experiments at realistic conditions. The experiments were performed at atmospheric pressure, and with constant atmospheric pressure in the region outside the pre-chamber, so that complications due to the rising chamber pressure in an engine are not present. This, together with the use of an optically-accessible pre-chamber, allows a clearer exploration of the structure of the ignition process, as a first step before the full engine ignition process can be understood.

METHODS

Experimental Methods

Apparatus

The apparatus used is shown in Fig. 1 (upper). It consists of a quartz cylinder 32mm ID and 32mm long, constrained at both ends with flat, 3mm thick steel plates. This volume forms the “pre-chamber”. One of these plates has a hole (3mm or 6mm), with sharp edges, which forms the orifice (symbol d). The other plate has an opening fit with a solenoid valve, which is used for filling the pre-chamber with the mixture to be ignited. The equivalence ratio of the mixture inside is ϕ_1 , usually $\phi_1=1$ and the fuel is ethylene, but in some experiments methane has been used. Outside the orifice, an impinging flow is used of a fully premixed air/fuel mixture of equivalence ratio ϕ_2 and with steady velocity U set in the range between 1 and 5 m/s. This burner was one part of the opposed jet burner of Ref. [7], and provides a turbulent flowing mixture to be ignited (once the outer flow is ignited by the jet, a flat turbulent premixed flame may be established parallel to the plate, which is the canonical stagnating premixed flame configuration). Although this is not an enclosed flow, we may loosely

use the term “chamber” in this paper to distinguish the region outside the orifice from the “pre-chamber”, which refers to the volume inside the quartz cylinder. In some experiments, $\phi_2=0$ so that the nature of the jet is easier to discern without the added complication of ignition of the outer fluid. Ignition is provided by a spark created by a focused 532 nm laser beam from a Continuum Surelite II Nd:YAG laser at 50mJ/pulse. Single shot operation allows for individually controlled and timed 7 ns long pulses to be delivered to the cylindrical, quartz pre-chamber where a 50 mm focal length lens focuses the beam to about 50 μm creating a spark at the center of the chamber due to local ionization of the fluid. This creates the ignition basis kernel with a toroidal shape characteristic of laser ignited mixtures [8,9].

Diagnostics

High speed chemiluminescence imaging of the OH^* and CH^* radicals were performed separately at 5.34 kHz with a Photron SA.1 and Lavision HS-IRO high speed intensifier. The imaging system was operated with an exposure time of 183 μs and an intensifier gain of 67. The imaging region captured both the optically accessible pre-chamber and the exiting jet into the ambient. Due to the large difference in signal contrast between the pre-chamber and jet chemiluminescence, an OD 0.1 neutral density filter was used to reduce chemiluminescence signal from the chamber. The imaging field-of-view was 89mm wide by 102mm tall with a resolution of 100 $\mu\text{m}/\text{pixel}$. Various movies of OH^* and CH^* chemiluminescence for both methane and ethylene and for various ignition locations, equivalence ratios, orifice sizes, and outer flows have been collected. In this paper, a few of these results are shown in the form of extracted stills, with the aim to reveal the location and shape of the reaction zone at various stages of the flame development.

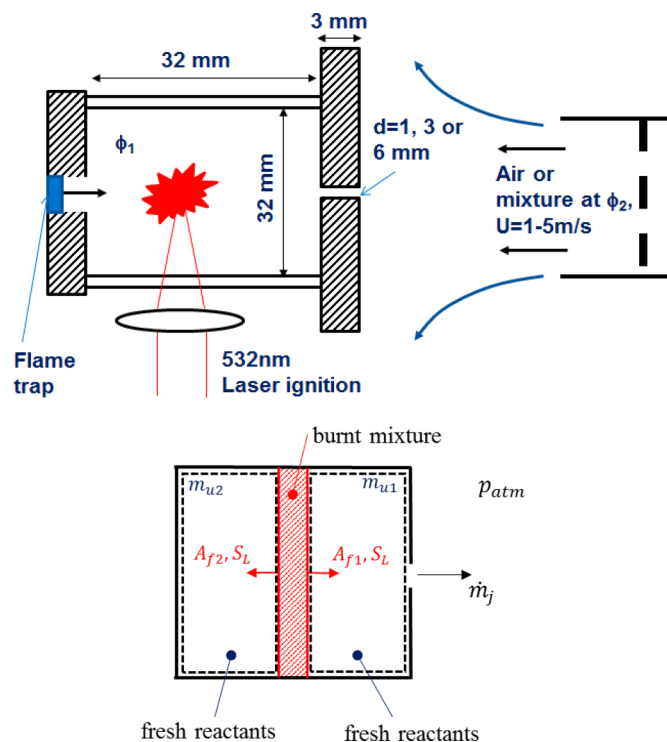


Figure 1. Upper: Sketch of the experimental rig and the flow pattern outside the pre-chamber. Lower: Definition of the thermodynamic systems for 0D modelling.

Modelling

Thermodynamic Approach

We follow an approach similar to the one proposed by Bradley and Mitchelson [10,11], who developed a system of volume-averaged governing equations for the pressure and mass inside an explosion vessel fit with a burst diaphragm, equivalent to treating it as a control volume with mass escaping. Considering an initial mixture of fresh reactants at a given equivalence ratio, the volume inside the pre-chamber is divided into regions with fresh reactants and combustion products. At each time step, the generation rate of the mass of burnt mixture is computed as:

$$\dot{m}_b = \rho_u A_f S_L \quad (1)$$

where ρ_u is the density of the unburnt mixture, A_f is the flame surface area and S_L is the laminar flame speed. The latter is computed using a correlation that allows the dependence on the pressure and temperature to be taken into account [12]. The computation of the flame surface area is one of the most important modelling aspects and the overall accuracy of the model strongly depends on the assumptions made for its computation. The flame shape can be either assumed a-priori (for example spherical flame shape as is done in Ref. [10]) or derived from CFD simulations. For the sake of simplicity, in this work a planar flame normal to the axis of the nozzle and initially located at a distance x_f (ignition location) from the back of the pre-chamber is considered. Two propagating fronts depart from the ignition location, one moving towards the nozzle and the other one in the opposite direction. In this way, the unburnt volume is divided into two different sections, separated by the flame. The two volumes of fresh reactants are considered as separate zones, as schematically shown in Fig. 1 (lower). Similar equations are applied to both zones excepting that the mass conservation of the unburnt mixture on the nozzle side has also an additional term to consider the mass flow coming out of the nozzle. The total flame area is equal to the sum of the surface area of the two flame fronts. A flame front disappears when it reaches either the nozzle or the back of the pre-chamber. The flame inside the chamber disappears when the mass of the fresh mixture is completely consumed or has exited the volume. The resulting system of equations is described next.

The mass conservation for the unburnt mixture on the two sides of the flame, with the control volume confined within the pre-chamber as indicated by the dashed lines in Fig. 1, is expressed as:

$$\frac{dm_{u1}}{dt} = -(\dot{m}_j + \dot{m}_{b1}) \quad (2)$$

$$\frac{dm_{u2}}{dt} = -\dot{m}_{b2} \quad (3)$$

where $\dot{m}_j = C_D \dot{m}_{id}$ is the mass flow rate exiting the orifice, computed from the ideal flow rate for compressible flows (determined by the pressure downstream of the nozzle and the total conditions upstream) by assuming a discharge coefficient C_D (C_D equal to 0.7 has been used in this study). Ideal gas behavior is assumed in all the zones

with thermodynamic properties evaluated as a function of temperature through NASA polynomials. In the unburnt mixture, transformations are assumed to be isentropic and density and temperature are computed by means of the equation of state and the isentropic relation (applied to the two unburnt zones separately):

$$p \rho_{u1,2}^{-\gamma_u} = K \quad (4)$$

where K is a constant determined from the initial conditions and γ represents the specific heat ratio. Starting from the value of the mass and density, the volume of each unburnt zone can be computed. The properties of the burnt mixture are determined by describing the combustion process as subsequent steps consisting of mixing between fresh reactants (mass determined from Eq. 1), combustion at constant volume and isentropic expansion to the pressure of the chamber. The energy released by the combustion process is computed from the lower calorific value of the fuel. A premixed mixture with a given equivalence ratio, temperature and pressure equal to the value downstream of the orifice (p_{atm}) is used as initial condition. The simulation is performed by imposing a constant time step Δt . At each time step sub-iterations are performed by imposing a pressure and computing the volume of the various zones. The convergence criterion is based on the volume of the mixture. If the total volume of the mixture (sum of the volume of burnt and unburnt zones) is not equal to the volume of the chamber the pressure is corrected until convergence is reached [10]. From the volume of the various zones and the area of the cross section, it is also possible to compute the location of the two flame fronts. The burnt mixture starts to flow out of the pre-chamber when the flame front propagating towards the nozzle reaches the orifice (i.e. no unburnt mass m_{u1} is left in the volume).

It is important to point out that more advanced models can also be built. As it will be evident from the following discussion, the shape of the flame depends on the ignition kernel and is usually far from two planar flames. Furthermore, the mixture exiting the orifice can be a mixture of both fresh reactants and products. These features can be included in a zonal model [13], however a proper calibration using experimental measurements and CFD simulations is required. Although a more detailed model is able to improve the quantitative predictions, the simplified description of the process provides useful qualitative trends.

Large Eddy Simulation

The openFOAM package (version 2.3) has been used for modelling the methane flames and for the 3mm orifice. The sub-grid scale stress tensor was modelled with the constant Smagorinsky model. The combustion model proposed by Weller et al. [14] was used. This model is based on the solution of the filtered regress variable together with a transport equation for the sub-grid flame wrinkling. Although the premixed flame model used here may not fully account for flame quenching due to heat loss or high stretch, and hence may not reveal fully the details of the flow and flame at the nozzle, the simulations provide some basic flow patterns during the flame expansion process. The computational domain includes the ignition chamber and extends downstream of the orifice for about 8d. The domain was discretized with a hexahedral mesh of about 0.5 million cells. Adiabatic no-slip