The impingement characteristics of an early injection GDI engine: a numerical study

This item was submitted to Loughborough University's Institutional Repository by the/an author.


Additional Information:

- This is the accepted manuscript version of the paper.

Metadata Record: [https://dspace.lboro.ac.uk/2134/22008](https://dspace.lboro.ac.uk/2134/22008)

Version: Accepted for publication

Publisher: © The Authors. Published by Sage.

Rights: This work is made available according to the conditions of the Creative Commons Attribution-NonCommercial-NoDerivatives 4.0 International (CC BY-NC-ND 4.0) licence. Full details of this licence are available at: [https://creativecommons.org/licenses/by-nc-nd/4.0/](https://creativecommons.org/licenses/by-nc-nd/4.0/)

Please cite the published version.
Corresponding Author:
Nicholas J Beavis, Department of Aeronautical and Automotive Engineering, Loughborough University, Loughborough, Leicestershire, LE11 3TU, UK
Email: n.beavis@lboro.ac.uk

THE IMPINGEMENT CHARACTERISTICS OF AN EARLY INJECTION GDI ENGINE: A NUMERICAL STUDY

Nicholas J Beavis¹, Salah S Ibrahim¹, Weeratunge Malalasekera²
¹Department of Aeronautical and Automotive Engineering, Loughborough University, Loughborough, UK
²School of Mechanical and Manufacturing Engineering, Loughborough University, Loughborough, UK

ABSTRACT
This paper describes the use of a Lagrangian discrete droplet model to evaluate the liquid fuel impingement characteristics on the internal surfaces of an early injection gasoline direct injection (GDI) engine. The study focuses on fuel impingement on the intake valve and cylinder liner between start of injection (SOI) and 20° after SOI using both a single- and multi-component fuel. The single-component fuel used was iso-octane and the multi-component fuel contained fractions of iso-pentane, iso-octane and n-decane to represent the light, medium and heavy fuel fractions of gasoline, respectively. A detailed description of the impingement and liquid film modelling approach is also provided.
Fuel properties, wall surface temperature and droplet Weber number and Laplace number were used to quantify the impingement regime for different fuel fractions and correlated well with the predicted onset of liquid film formation. Evidence of film stripping was seen from the liquid film formed on the side of the intake valve head with subsequent ejected droplets being a likely source of unburned hydrocarbons and particulate matter emissions. Differences in impingement location and subsequent location of liquid film formation were also observed between single- and multi-component fuels. A qualitative comparison with experimental cylinder liner impingement data showed the model to well predict the timing and positioning of the liner fuel impingement.

INTRODUCTION

The Gasoline Direct Injection (GDI) engine has a number of practical advantages over the more traditional port-fuel injection strategy including, eliminated transient dwell time, improved fuel metering for reduced unburned hydrocarbon (UHC) emissions, reduced knock propensity due to charge cooling effects and the potential for significant fuel economy improvements with stratified charge combustion strategies. However, since the sustained interest in GDI technology in the early 1990’s a number of challenges have been the subject of continued research in support of fully exploiting the advantages of the GDI engine.

Whilst late injection stratified charge injection strategies offer significant fuel economy benefits, the complexity of regulating the fuel-air mixture around the spark plug at the point of spark timing remains a challenge and many manufacturers still adopt an early injection homogeneous charge injection strategy at both low and high load operating conditions. A challenge with an early injection strategy lies in the avoidance of excessive impingement on the piston, liner or intake valve with low in-cylinder pressures and high fuelling quantities. Liquid fuel impingement is linked with particulate matter (PM) formation and unburned hydrocarbons (UHC).

The use of detailed numerical techniques to model the fuel injection process has been common place for a number of years now but studies typically utilise a single component fuel to represent the reality of a complex multi-component fuel, largely due to computational restrictions. Advantages to modelling multiple components of the fuel are: (1) improvements in the modelling of droplet evaporation due to it being dependent on the different volatilities of the individual components within the fuel rather than a single component, and (2) the improvement in transient droplet behaviour due to droplet mass, momentum and subsequent interactions with the continuous phase, being more representative of the physical processes that occur during fuel injection in an engine.
This leads to improvements in reliability of impingement and liquid film numerical predictions.

This study evaluates the impingement characteristics in an early injection GDI engine. Over the crank angle range modelled (SOI to SOI+20°), the study focuses on impingement of the intake valve head and cylinder liner wall. It compares the results of a single-component and multi-component fuel modelling approach with respect to differences in impingement characteristics and the liquid film formed. A detailed description of the impingement and liquid film modelling approach is also given, providing a methodology for identifying how the droplet impingement regime changes over the injection process.

THE EXPERIMENTAL SETUP

Engine and Injector

The experimental data used to validate the computational results was gathered from a quiescent chamber and a single cylinder four stroke motored optical research engine designed and built by the advanced powertrains group at Jaguar Cars. Key information is summarised in Table 1.

The injector is a vertically and centrally mounted six-hole injector with y-plane plume symmetry. Plumes 3/4 and 2/5 were designed with large injection angles to avoid liner impingement and penetrate deep into the combustion chamber, whilst plumes 1/6 were designed with a shallow injection angle to direct the fuel around the spark plug and to generate a rich stratified fuel-air mixture when utilising a late injection strategy. The spray plume orientation is shown in Figure 1.
Figure 1 – Spray plume orientation, reproduced from 7
Table 1 – Experimental configuration

Operating Conditions

The model is validated against published experimental data at a standardised motored condition as depicted in Table 2.
<table>
<thead>
<tr>
<th>Engine Speed</th>
<th>1500 rpm</th>
</tr>
</thead>
<tbody>
<tr>
<td>Barometric Pressure</td>
<td>1 bar</td>
</tr>
<tr>
<td>Intake Manifold Pressure (abs)</td>
<td>0.5 bar</td>
</tr>
<tr>
<td>Intake Air Temperature</td>
<td>301 K</td>
</tr>
<tr>
<td>Exhaust Backpressure (abs)</td>
<td>1.016 bar</td>
</tr>
<tr>
<td>Exhaust Temperature</td>
<td>784 K</td>
</tr>
<tr>
<td>Fuel</td>
<td>Iso-Octane</td>
</tr>
<tr>
<td>SOI</td>
<td>80 °ATDC\text{INT}</td>
</tr>
<tr>
<td>Injection Pressure</td>
<td>150 bar</td>
</tr>
<tr>
<td>Pulse Width</td>
<td>0.78 ms</td>
</tr>
</tbody>
</table>

Table 2 - Experimental operating condition

THE NUMERICAL MODEL

The numerical model was developed using the CFD code STAR-CD v4.22. The model validation with respect to the in-cylinder flow field has been documented previously thus for brevity will not be discussed here. A validation of the fuel injection model is presented in the ‘Results and Discussion’ section. The numerical model, initial and boundary conditions and physical sub-models used are discussed further below.

The Computational Domain

The computational domain is illustrated in Figure 2. A cell size dependency study was completed as part of the model validation process (results not shown here). It was found that a cell size of 0.7-0.8mm provided an approximately mesh independent solution for plume tip penetration, spatial mixture fraction variation and droplet size distribution.
Figure 2 – Computational mesh representing the single cylinder optical research engine

Initial and Boundary Conditions

The simulation was initialised at 80°ATDC using the results from a cold-flow single-phase simulation. Numerical wall temperatures were set based on the experimental conditions or approximated based on surrounding material and gas temperatures. The boundary conditions are summarised in Table 3.
Intake Plenum and Port Wall Temperatures | Adiabatic
--- | ---
Intake Valve Temperatures | 323 K
Exhaust Valve Temperatures | 363 K
Cylinder Head Temperature | 363 K
Liner Temperature | 293 K
Piston Temperature | 301 K
Intake Gas Pressure | 528 mbar
Intake Gas Temperature | 301 K

Table 3 – Numerical boundary conditions

The Fuel Injection Model

Droplet Distribution | Rosin-Rammler: $X = 14 \times 10^{-6}$ m, $q = 2.3$
--- | ---
Fuel Temperature | 363 K
Number of Injected Parcels | 50’000 parcels per jet
Droplet Initial Velocity | Shown in Figure 3
Injection Rate | Shown in Figure 3
Total Injected Mass | 13.8 mg

Table 4 – Fuel Injection Inputs
A Rosin-Rammler distribution was used to provide an initial droplet size distribution to the injected parcels. The constant ‘q’ (the ‘shape’ parameter) was set to 2.3 based on the experimental works of 9 which used a similar injector configuration and experimental conditions. The constant ‘X’ (the ‘scale’ parameter) was set to $14 \times 10^{-6}$ m which provided the best match against experimental Phase Doppler Anemometry (PDA) droplet size data.

A dependency study was completed to evaluate the influence of the number of injected parcels on the plume tip penetration and average droplet characteristics. The results (not shown here) showed that with the current mesh and sub-models, 50’000 parcels per jet provided a parcel number independent solution with acceptable computational expense.

The steady-state injection mass flow rate was measured during a previous experimental study 10 as 16.68 g/s but the time varying mass flow rate profile for the injector was not available. To ensure a reasonable rate profile was supplied to the model, the opening and closing injection rate characteristics from a similar injector were combined with the known steady-state flow rate to create a realistic injection profile. The final profile is shown in Figure 3.

The droplet velocity at the injector nozzle was imposed via a time-dependent profile as shown in Figure 3, with an increase in initial droplet velocity used around the needle opening and closing to provide the best match against experimental plume tip velocity data – results shown below in the section ‘Fuel Injection Model Validation’.
In this study both a single-component fuel and a multi-component fuel are modelled. The fuel component and respective initial mass fraction is defined in Table 5. The multi-component fuel is modelled as a miscible mixture where the evaporation of each component is dependent on the concentration and vapour pressure of the other components in the mixture. Since the components modelled in this study have a similar molecular structure (i.e. are all Alkanes), the application of Raoults’s law to obtain the vapour mole fraction at the droplet/liquid surface was deemed an acceptable approximation. A comparison of the vapour pressures for each fuel component and the multi-component fuel (calculated by the sum of the partial pressures for each component) against a typical gasoline fuel is provided in Figure 4. All fuel properties required by the CFD code are extracted from NIST tables for the respective component but a sample of key fuel properties are shown in Table 6.
<table>
<thead>
<tr>
<th>Single-Component Fuel</th>
<th>Multi-Component Fuel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Component Fuel</td>
<td>Mass Fraction</td>
</tr>
<tr>
<td>Iso-Octane</td>
<td>1</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Table 5 – Definition of fuels used

Figure 4 – Comparison of fuel vapour pressures, including the multi-component fuel against a typical gasoline fuel.
<table>
<thead>
<tr>
<th>Fuel Property</th>
<th>Iso-Pentane</th>
<th>Iso-Octane</th>
<th>n-Decane</th>
<th>Gasoline</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density (20°C) [kg/m³]</td>
<td>616</td>
<td>691.9</td>
<td>730</td>
<td>719</td>
</tr>
<tr>
<td>Dynamic Viscosity (20°C) [g/m-s]</td>
<td>0.233</td>
<td>0.5</td>
<td>0.92</td>
<td>0.3-0.8</td>
</tr>
<tr>
<td>Surface Tension (20°C) [mN/m]</td>
<td>16.05</td>
<td>18.77</td>
<td>23.83</td>
<td>25.8</td>
</tr>
<tr>
<td>Boiling Point [K]</td>
<td>301.1</td>
<td>372.2</td>
<td>447.2</td>
<td>303-463</td>
</tr>
<tr>
<td>Critical Temperature [K]</td>
<td>460.4</td>
<td>543.9</td>
<td>617.7</td>
<td>544-562</td>
</tr>
<tr>
<td>Critical Pressure [Bar]</td>
<td>33.76</td>
<td>23.88</td>
<td>21.03</td>
<td>25.7-32.6</td>
</tr>
<tr>
<td>Enthalpy of Vapourisation [kJ/mol]</td>
<td>25.0</td>
<td>35.1</td>
<td>47.4</td>
<td>35.4-37.3</td>
</tr>
</tbody>
</table>

Table 6 – Fuel Properties

The Physical Sub-Models

A detailed discussion on the physical sub-models used in the numerical model is omitted for brevity but summarised in Table 7. This is with exception to the droplet-wall interaction and liquid film modelling approaches which will be discussed in detail.
Turbulence (gas phase) | RNG k-ε $^{12,13}$  
---|---
$C_u=0.085$, $C_{z1}=1.42$, $C_{z2}=1.68$, $C_{z3}=1.42$, $C_{z4}=-0.387$, $k=0.4$, $β=0.012$, $η_0=4.38$, $σ_k=0.719$, $σ_0=0.719$, $σ_h=0.9$, $σ_n=0.9$

Turbulence – Near Wall | Angelberger $^{14}$  
---|---
$y^+_sw=13.2$, $a_w=2.075$, $b_w=3.9$

Turbulence Dispersion | Gaussian pdf $^{15}$

Breakup Model | Pilch & Erdman $^{16}$  
---|---
$B1=0.375$, $B2=0.2274$

Collision Model | O’Rourke $^{17}$, with addition sub-models for algorithm speed-up $^{18}$, automatic coalescence timestep adjustment $^{19}$ and additional geometric constraints $^{20}$  
---|---
$K_{km}=1$

Droplet-Wall Interaction Model | Senda et al. $^{21-25}$, Bai & Gosman $^{26}$, Rosa et al $^{27}$  
---|---
$c_f=0.7$

Leidenfrost temperature determination | Habchi $^{28}$ & Spiegler $^{29}$

Liquid Film Model | Bai & Gosman $^{30}$  
---|---
$γ_c=0.8$

Liquid Film Model – Boiling Model | White $^{31}$  
---|---
$C_{sf}=0.06$, $n=3$, $C_s=1.2$, $c_{max}=0.15$, $c_{min}=0.09$

Liquid Film Model – Film Stripping due to Flow Over Edge | Friedrich $^{32}$  
---|---
$θ_{min}=45°$, $FR_c=1$, $c_1=3.78$, $q=1.5$

Liquid Film Model – Film Stripping due to Wave & Body-Force Induced Instability | Fourcart $^{33}$

Liquid Film Model – Effect of Contact Angle | Fourcart $^{33}$  
---|---
$θ_c=35°$, $c=1$

### Table 7 – Summary of physical sub-models and constants used

### Droplet-Wall Interaction Modelling

The study of the interaction between impinging droplets and solid surfaces also continues to be an area of active research in both experimental and numerical fields.
The work of Bai and Gosman is one of the first to summarise and propose seven characteristic regimes of droplet impingement: stick, rebound, spread, boiling induced break-up, rebound with breakup, breakup, splash.

Through this work, Bai and Gosman also proposed a model to represent the droplet impingement process. Their model is based on mass, momentum and energy conservation constraints and the addition of a randomising procedure on the post-impingement characteristics to represent the stochastic nature of the impingement process. This modelling approach has been furthered by a number of researchers including that of Rosa et al. and the works of Senda et al. The impingement model used in this study is primarily based on the works by Senda et al. but also incorporates many of the features of the model by Bai and Gosman and Rosa et al. This modelling approach provides a significant reduction in the number of user defined constants which is beneficial with limited experimental data for model validation.

The droplet impingement model defines three regimes based on temperature $T^*$ which is defined by:

$$T^* = \frac{T_W - T_N}{T_L - T_N} \tag{1}$$

Where:
- $T_W$ is the wall temperature
- $T_L$ is the Leidenfrost temperature
- $T_N$ is the Nukiyama temperature and is defined by:

$$T_N = B_S T_{sat} \tag{2}$$

B_S is a user defined constant and set as 1 in this study. $T_{sat}$ is the liquid saturation temperature dependent on the liquid vapour pressure which can either be calculated via the Clausius-Clapeyron equation or by NIST tables, as is the case in the CFD code used in this study.

Imposing a reasonable estimate of the Leidenfrost temperature (which is difficult to measure experimentally) is critical for accurate predictions of high temperature wall-wetting and the modelling approach used in this study is discussed further in the section below.
The droplet Weber number (We) and Laplace number (La) are used throughout the impingement model are defined by:

\[
We = \frac{\rho D U^2}{\sigma}
\]

\[
La = \frac{\rho \sigma L}{\mu^2}
\]

Where:
- \(\rho\) – droplet density
- \(D\) – droplet diameter
- \(U\) – droplet velocity
- \(\sigma\) – droplet surface tension
- \(\mu\) – droplet dynamic viscosity

**Regime 1: Free convection and nucleate boiling regime, \(T^* \leq 0.00\)**

This range is sub-divided into conditions for a dry surface and wetted surface.

**Dry Surface:**
- Drop-drop interaction or deposition: \(We \leq 400\)

In this regime the interaction between droplets during impingement affects their residence time on the surface, spreading, and droplet and film stability. After impact, droplet interactions are defined based on collision (including any coalescence) and secondary breakup models. Droplet deposition occurs until the surface coverage ratio is exceeded and a liquid film is formed – discussed further in the section on "Liquid Film Modelling".

**Splash:** \(We > 400\)

**Wetted Surface:**
- Drop-film interaction: \(We \leq 300\)

At low Weber numbers, three film breakup sub-regimes are defined from the experimental works of \(^{34,35}\), as a function of the non-dimensional film thickness...
(δ) and consequently define the child droplet diameter ratio and Weber number [21].

- Rim type: breakup or droplet ejection of one of a few droplets at the outer edge of the film
- Cluster type: breakup into clusters of many small droplets
- Column type: breakup into one or a few droplets from a column of rising fluid, formed from the resulting surface waves reflecting back to the point of impact

Deposition/joins existing film: $300 < We < W_{ecr}$

Splash: $We \geq W_{ecr}$

The critical droplet Weber number, $W_{ecr}$ is defined by:

$$W_{ecr} = (2164 + 7560\delta^{1.78})La^{-0.2} \tag{5}$$

Where:

$$\delta$$ is the non-dimensional film thickness, $\delta = (2/3)/\beta^2 \tag{6}$

$$\beta$$ is the droplet spreading factor, $\beta = 0.87(We/6+2)^{0.5} \tag{7}$

The droplet spreading factor is used to relate the impinging droplet Weber number to the child droplet outgoing Weber number and diameter, and in the calculation of the critical Weber number for determining if a high Weber number impinging droplet is deposited into the film or splashes [21].

Regime 2: Transition boiling regime, $0.00<T^*\leq1.00$

- Rebound: $We \leq 200$
- Spread/deposition: $200 < We \leq W_{ecr}$
- Splash: $We > W_{ecr}$

Regime 3: Film boiling regime, $T^*>1.00$

- Rebound: $We \leq 100$
Leidenfrost Temperature Modelling

In this study, the model proposed by Habchi is used for estimating the condition specific Leidenfrost temperature and is defined by:

\[ T_{cr} = T_{sat} + \Delta T \]  \hspace{1cm} (8)

Where:

\[ T_{cr} \] represents the pressure dependent Leidenfrost temperature, \( T_L \)
\[ \Delta T = \begin{cases} T_{cr|1 \text{ bar}} - T_b & : \text{if } (p \leq 1 \text{ bar}) \\ \left( \frac{T_{cr|1 \text{ bar}} - T_b - A}{T_e - T_b} \right) (T_c - T_{sat}) + A & : \text{if } (p > 1 \text{ bar}) \end{cases} \]  \hspace{1cm} (9)

\( T_{cr|1 \text{ bar}} \) is the Leidenfrost temperature at 1 bar gas pressure
\( T_b \) is the normal boiling temperature of the liquid at 1 bar gas pressure
\( T_c \) is the critical temperature of the liquid
\( A \) is calculated via: \( A = \text{Max}(1, T_{cr|1 \text{ bar}} - T_c) \)

Hence the Habchi model assumes that the Leidenfrost temperature is static below atmospheric pressure, but at pressures greater than atmospheric it tends linearly towards \( A \) when the gas pressure tends towards the critical pressure.

As mentioned previously, determining the Leidenfrost temperature experimentally is difficult and a great deal of variation is seen in published data. To allow determination of Leidenfrost temperature at 1 bar gas pressure across a variety of fuels for which reasonable experimental data may not exist, the model proposed by Spiegler et al. is used. Spiegler et al. suggest that at conditions where the gas pressure is significantly less than the critical pressure, the result of \( \frac{27}{32} T_c \) provides a good approximation of the foam limit and hence the minimum of the heat flux versus temperature curve at standard conditions. Thus the Leidenfrost temperature at 1 bar gas pressure can be approximated by:

\[ T_{L|1 \text{ bar}} = \frac{27}{32} T_c \]  \hspace{1cm} (10)
Where $T_c$ is determined from NIST tables for the fuel.

It is worth noting that in reality, the Leidenfrost temperature is a dynamic property also dependent on the impinging droplet conditions. The Leidenfrost temperature model used was based on sessile droplets and does not account for the impact of droplet dynamics on Leidenfrost temperature.

**Liquid Film Modelling**

A liquid film or droplet attached to a wall is subject to the following major physical phenomena:

- Weight of the liquid film or droplet
- Surface tension
- Liquid-gas and wall-liquid shear stress
- Imparted momentum from impinging droplet and the surrounding gas phase and lost momentum due to splashing
- Flow separation and sheet breakup
- Heat transfer: convection to the surrounding gas, conduction to the solid surface
- Evaporation to the surrounding gas

In this study, liquid films are modelled using an Eulerian approach based on the work of Bai and Gosman, but a Lagrangian approach is incorporated for the modelling of individual impinged droplets up to the surface coverage ratio limit. After a droplet has been determined to be deposited, it is assumed to spread into a cylindrical form with diameter $D_s$. The surface coverage ratio, defined by Equation (11), is constantly evaluated as droplets impinge on the solid surface until exceeding a predefined value (0.8 in this study), after which the droplet parcels on the cell face merge into a liquid film and are subsequently treated under the Eulerian approach of Bai and Gosman. A liquid film spreading into a new cell face will instantly absorb any individual impinged droplet parcels in the new cell face into the liquid film.

$$
\gamma_c = \frac{\pi}{4A_c} \sum_l D_{s,l}^2 N_l \tag{11}
$$

Where $A_c$ is the area of the cell face where the droplet parcel is located, $N_l$ is the number of droplets in the $l^{th}$ parcel.

Film stripping is defined by three different mechanisms.
- Stripping due to wave instability generated by adjacent flow
- Stripping due to body-force induced instability (e.g. gravity, piston acceleration)
- Breakup caused by flow over a sharp edge

The modelling approach for film stripping due to wave and body-force instabilities is adopted from Foucart et al. 33, whereas the approach suggested by Friedrich et al. 32 is used for breakup across a sharp edge.

RESULTS AND DISCUSSION

Fuel Injection Model Validation

The fuel injection model was first validated against available experimental data. An additional validation exercise of the liner impingement process is presented in a later section ‘Cylinder Liner Impingement’. Experimental results to validate the fuel injection model were available in the form of plume tip penetration, plume tip velocity and $D_{10}$ droplet diameter.

Plume tip penetration was extracted from $^3$ and compared against the predicted plume tip penetration for iso-octane (single-component fuel) as shown in Figure 5, showing good agreement between experimental results and numerical predictions.
Spray plume tip velocity was calculated from the derivative of the plume tip penetration data presented previously and is shown in Figure 6. Numerical predictions show good agreement with experimental plume tip velocity and well capture the steep rise in plume tip velocity soon after the start of injection. As a consequence, this provides excellent agreement between experimental and numerical results for tip penetration during the early stages of injection.
Experimental PDA data was extracted from the D_{10} droplet diameter was measured in a constant volume chamber for plume 2 at z=-25mm from the injector nozzle tip across a range of temperature and pressure conditions and compared against the numerical results and is shown in Figure 7. Unfortunately, experimental results were not available at the standardised condition of T_f=363K and 0.5bar gas pressure but the results suggest that the droplet diameter is within the expected range and the change in droplet diameter over time closely matches the experimental results providing increased confidence in the capability of the droplet breakup model to satisfactorily predict the secondary breakup processes.
Figure 7 – A Comparison of Experimental and Predicted $D_{10}$ Droplet Diameter at $z=-25\text{mm}$ from the injector tip for iso-octane

Impingement Regime Determination

Through both experimental studies \cite{4,5,39,40} and the current numerical study, three spray impingement locations have been identified as shown by Figure 8: intake valve, cylinder liner and piston.
Over the crank angle range evaluated in this study, two instances of impingement will be discussed in detail; impingement of plume 1 on the intake valve and impingement of plume 6 on the cylinder liner.

Before evaluating each impingement location in detail, it is useful to evaluate the expected impingement regime within the impingement model.

The regime within the impingement model is defined based on parameter $T^*$ (Equation (1)) which was calculated for the different fuel components and surfaces within the combustion chamber and is shown in Table 8.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Iso-Pentane</td>
<td>363</td>
<td>0.953</td>
<td>323</td>
<td>0.485</td>
<td>301</td>
<td>0.228</td>
<td>293</td>
<td>0.134</td>
</tr>
<tr>
<td>(Exhaust Valve and Head)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Iso-Octane</td>
<td>0.141</td>
<td>(Intake Valve)</td>
<td>-0.32</td>
<td>(Piston)</td>
<td>-0.574</td>
<td>(Cylinder Liner)</td>
<td>-0.666</td>
<td></td>
</tr>
<tr>
<td>n-Decane</td>
<td>-0.809</td>
<td></td>
<td>-1.338</td>
<td></td>
<td>-1.629</td>
<td></td>
<td>-1.735</td>
<td></td>
</tr>
</tbody>
</table>
Upon inspection of Table 8, it is clear that for the single-component fuel of iso-octane, droplets will be in the free convection and nucleate boiling regime ($T^*<0$) for impingement on the intake valve and cylinder liner (surface temperatures, $T_w$ of 323K and 293K respectively). For the multi-component fuel, the situation is more complex since each droplet is made up of a fraction of three components. Unfortunately, due to restrictions in the code, there is no method for determining the instantaneous fraction of a given component in each droplet and hence deduce the $T^*$ value of impinging droplets. That being said, it is expected that by 6°ASOI a significant fraction of iso-pentane will have evaporated from the injected droplets, and since the iso-pentane fraction is only 1/3 of the total mass fraction at the inception of each injected droplet, the majority of the droplets impinging on the intake valve will also fall within the free convection and nucleate boiling regime ($T^*<0$).

Impingement on the cylinder liner is far later in the injection process hence an even smaller quantity of the iso-pentane fraction is expected to remain and the majority of impinging droplets will also be within the free convection and nucleate boiling regime ($T^*<0$).

Based on the range of values for $T^*$, it is clear that no droplet in either the single- or multi-component fuel cases will impinge in the film boiling range ($T^*>1$).
Intake Valve Impingement

From results shown in Figure 9 and using equations (5), (6) and (7), it is possible to calculate the critical Weber number (We$_{cr}$) at a number of key points within the impingement process, shown in Table 9. Due to the impinging droplets covering a range of Weber numbers and Laplace numbers, the critical Weber number was calculated at approximate upper and lower values to provide an expected range of We$_{cr}$.

<table>
<thead>
<tr>
<th>Time [*ASOI]</th>
<th>Comments</th>
<th>We</th>
<th>La</th>
<th>We$_{cr}$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Upper</td>
<td>Lower</td>
<td>Upper</td>
</tr>
<tr>
<td>5</td>
<td>Pre-impingement</td>
<td>5000</td>
<td>1200</td>
<td>524</td>
</tr>
<tr>
<td>6</td>
<td>Immediately post initial impingement</td>
<td>3000</td>
<td>2000</td>
<td>1000</td>
</tr>
<tr>
<td>9</td>
<td>Immediately prior to film formation</td>
<td>1200</td>
<td>600</td>
<td>900</td>
</tr>
<tr>
<td>10</td>
<td>First crank angle of film formation</td>
<td>1000</td>
<td>600</td>
<td>800</td>
</tr>
<tr>
<td>13</td>
<td>Splitting of plume due to impingement on top of valve head</td>
<td>1200</td>
<td>600</td>
<td>700</td>
</tr>
</tbody>
</table>

Table 9 – Table shows the range of droplet Weber and Laplace numbers and calculated We$_{cr}$ through the intake valve impingement process for droplets from the multi-component fuel
At 5° and 6° ASOI the Weber number of the impinging droplets are all significantly greater than the Weber number needed for the generation of a liquid film (We ≤ 400) hence splash off the surface.

By 9° ASOI, the Weber number of the impinging droplets has reduced considerably. The table above shows a lower Weber number of 600 but based on a velocity of 50 m/s, any droplet of ~4 μm or less will have a Weber number less than 400 and hence be deposited on the surface.

By 10° ASOI there are sufficient deposited droplets that the surface coverage ratio is exceeded and a liquid film has formed. Once a film has formed the regime for splashing or deposition is defined based on the critical Weber number W_{ecr} which, based on Table 9, will be in the region of 570 < W_{ecr} < 650. Thus once a liquid film has formed, it will quickly grow due to the larger Weber number criteria and greater proportion of smaller, lower Weber number droplets.

By 13° ASOI the descending intake valve causes the plume to be split by the edge of the valve head and droplets be deflected beneath and above onto the top surface of the valve.
head. As a consequence of the sudden reduction in velocity of droplets impinging the top surface, a liquid film is rapidly formed. Movement of the film across the surface also occurs due to imparted momentum by the spray.

There is evidence of a large number of droplets being stripped from the liquid film due to charge motion over the valve head edge, indicated by the large slow moving droplets shown in Figure 10 at 19°ASOI. This process is of significance due to the size and velocity of the new droplets; most in the region of 100-300μm but some >500μm, which will take a significant period of time to evaporate and very easily be a source of UHC and PM emissions.

![Figure 10](image)

**Figure 10** – Images illustrate the presence of very large and slow moving droplets formed underneath the valve head due to film stripping over the sharp edge of the intake valve head at 19°ASOI. Droplets in (a) are coloured by droplet diameter and in (b) are coloured by droplet velocity magnitude with the valve head outline shown in grey.

A difference of note between the single- and multi-component fuel predictions is the larger predicted film thickness for the multi-component fuel case. As shown in Figure 11 at 13°ASOI, the film thickness formed on the side of the intake valve in the single-component fuel case is predicted to be ~20microns but is predicted to be ~50microns with the multi-component fuel. This is due to the presence of the heavier n-decane fraction in the multi-component fuel reducing the number of droplets that have
evaporated prior to impingement, thus increasing the number of droplets reaching and impinging on the intake valve and available to contribute to the liquid film.

Another observed difference between the predictions for the multi-component fuel case and single-component fuel case is the location of impingement. Figure 11-(a) and (b) compare the impinging droplets at 14°ASOI. Recall Figure 9 at 6°ASOI for the multi-component fuel, the spray plume enters the visualised domain at approximately $z=-7.2\text{mm}$ whereas by 14°ASOI (Figure 11-(b)) the plume enters the domain at approximately $z=-6.8\text{mm}$, displaced towards the cylinder head by the in-cylinder charge motion. This is thought to be due to the presence of the lighter iso-pentane fraction within the multi-component fuel, allowing the droplets to be more easily influenced by the momentum of the in-cylinder flow field. The single-component fuel (Figure 11-(a)) in contrast, due to only containing a single component of iso-octane, is less influenced by charge motion. The onset from this is that towards the end of injection process the multi-component fuel predicts a greater proportion of fuel to be injected directly onto the back on the intake valve head, causing a difference in film formation between the two cases. Figure 11-(c) and (d) show the film thickness for single- and multi-component fuels. It is clear that in the case of the multi-component fuel, a larger and thicker liquid film is formed on the top of the valve head compared with the single-component fuel. Thus the prevalent film stripping and child droplet formation processes will be different for the single- and multi-component fuel cases; predominantly stripping over a sharp edge for the single-component fuel and a greater number of child droplets formed via flow induced wave instabilities over the valve head surface for the multi-component fuel. This will impact the droplet size distribution and subsequent fuel-air mixture through the remainder of the intake and compression strokes.
Cylinder Liner Impingement

Prior to investigating the cylinder liner liquid film development, a validation exercise was completed on the cylinder liner impingement process.
Three different pieces of experimental data were used to support this validation exercise: (1) quantitative data from a heat flux sensor study and (2) qualitative mean spray images were used to validate the timing and location of impingement, and (3) a high resolution instantaneous spray image was used to help validate the subsequent plume tip motion following liner impingement.

Figure 12 reproduces the pertinent results from the heat flux sensor study where a heat flux sensor was placed at consecutive positions around the periphery of the cylinder bore at approximately 17mm below the head gasket plane. The resultant data from the heat flux sensor provides information on the predominant impingement location and timing for plume 6. Upon inspection of these experimental results, it is clear that with iso-octane and for a fuel temperature ($T_f$) of both 363K and 293K, plume 6 predominantly impinges the liner between 30°-50° to the horizontal and at approximately 15°ASOI.

Note: Data at $T_f=293K$ was included in Figure 12 since it indicates that the impingement location and timing varies little between $T_f=293K$ and $T_f=363K$ which allows comparison of the predicted results with the second and third experimental images with more confidence, since these images are only available at $T_f=293K$.

The second piece of experimental data used to support model validation is shown in Figure 13 (a & c) and are mean spray images along the swirl plane at z=-15mm from the head gasket plane, at 13°ASOI (identified as the timing of first impingement) and 15°ASOI. Figure 13 (b & d) show results from the numerical simulations for single-component fuel iso-octane at the same crank angles and cutting plane. The numerical results are shown to agree well with experimental data sets in Figure 12 and Figure 13 (a & c), correctly predicting plume 6 to impinge the cylinder liner at 30°-50° to the horizontal and having just impinged the liner by 13°ASOI.

The third piece of experimental data is extracted from and shown in Figure 14. Here a comparison is made between an instantaneous swirl plane experimental image and the single-component fuel predictions to evaluate the post impingement droplet trajectory. This comparison also indicates good representation of the impingement characteristics of the spray and subsequent circumferential motion of the spray around the liner.

More complex impingement regimes were also identified in the experimental data, including multiple roll up vortices of order 5mm diameter interacting with the spray just behind the plume tip. Predicted results were found not to capture the spray dynamics at this level of detail and would need very high resolution of the turbulent flow structures only possible with significant localised mesh refinement.
Figure 12 – (a) Indicates the peripheral sensor locations for the heat flux sensor taken from\(^4\) (b) Shows peak heat flux and the equivalent crank angle at various peripheral location for iso-octane and \(T_f=363\text{K}\) and \(T_f=293\text{K}\), reproduced using results from\(^4\).
Figure 13 – Figure compares the location of liner impingement for plume 6 with iso-octane. (a & c) Experimental images\textsuperscript{5} show mean (60 cycle ensemble-average) spray development with iso-octane at $T_f=293K$, 0.5bar intake pressure, 1500rpm, illuminated by an applied laser sheet due to Mie-scattering and the pixel intensity coloured for improved visualisation. An overlay is applied to more easily compare the impingement locations with numerical results. Red arrows in the experimental images indicate the spray plume crossing the laser sheet and impingement on the liner. (b & d) Predicted droplets for the single-component fuel, coloured by droplet velocity magnitude and the cylinder liner indicated by a black circle.
Figure 14 – Figure compares liner impingement dynamics for plume 6 at 15°ASOI. (a) Experimental image\textsuperscript{5} is an instantaneous spray image for iso-octane at T\textsubscript{r}=293K, 0.5bar intake pressure, 1500rpm, illuminated by an applied laser sheet due to Mie-scattering, indicating radial droplet motion post-liner impingement. (b) Predicted droplets for the single-component fuel, coloured by droplet velocity magnitude and with x-y plane velocity vector arrows, the cylinder liner indicated by a black circle. Note: images are not of equal scale.

Very similar impingement timing and location was seen between the single-component and multi-component fuels hence the remainder of the analysis on cylinder liner impingement will be using the results from the multi-component fuel.

Figure 15 illustrates the development of a fuel film on the liner surface between 18 and 19°ASOI for the multi-component fuel. Performing the same analysis as completed for the intake valve impingement, the droplet Weber number and Laplace number can be calculated and compared against the critical Weber number for film formation within the free convection and nucleate boiling regime of the impingement model. The results are shown in Table 10 and it is clear that most impinging droplets upto 18°ASOI are above the critical Weber number for deposition but by 19°ASOI the Weber number of impinging droplets has reduced sufficiently that a large proportion of impinging droplets are beneath the critical Weber number and are deposited, with the coverage ratio limit quickly exceeded and a liquid film formed.
The predictions indicate that a film thickness on the order of 5μm is formed on the liner surface. Drake et al. 41 completed a number of experimental tests using a refractive-index-matching approach to evaluate continuous cycle-by-cycle piston film development using both a pressure-swirl and multi-hole fuel injector. Their results for the multi-hole injector showed an area-averaged film height of up to 1μm, with images indicating a peak film thickness of 1-2.5μm, providing additional confidence that the liquid film predictions presented in this study are of the correct magnitude.

![Figure 15](image-url)

**Figure 15** – Development of a liquid film on the cylinder liner as a consequence of spray impingement with a multi-component fuel at 18° and 19°ASOI

<table>
<thead>
<tr>
<th>Time [°ASOI]</th>
<th>Comments</th>
<th>We</th>
<th>La</th>
<th>We_{cr}</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Upper</td>
<td>Lower</td>
<td>Upper</td>
</tr>
<tr>
<td>18</td>
<td>Pre-liquid film formation</td>
<td>1600</td>
<td>400</td>
<td>1000</td>
</tr>
<tr>
<td>19</td>
<td>Immediately post initial film formation</td>
<td>800</td>
<td>300</td>
<td>500</td>
</tr>
</tbody>
</table>

**Table 10** - Table shows the range of droplet Weber and Laplace numbers and calculated We_{cr} for two crank angles at the point of liner liquid film formation for the multi-component fuel
As seen during liquid film formation in intake valve impingement, a thicker liquid film is predicted with the multi-component fuel due to less droplets having evaporated prior to the spray reaching the cylinder liner.

It is expected that the use of multi-component fuels will provide improved predictions of droplet evaporation and liquid film formation and evaporation characteristics. This is particularly true when modelling firing cycles, where the elevated wall temperatures would still be below the saturation temperature of the heavier components of a multi-component fuel allowing deposition; whereas the wall temperatures would be above the saturation temperature of a typical single component surrogate like iso-octane, thus not providing realistic liquid film predictions. This will also allow improvements in the modelling of subsequent processes such as pool-fires, oil dilution and entrainment of large particles that can contribute to UHC and PM emissions.

CONCLUSIONS

A numerical model has been developed to quantify the impingement characteristics in an early injection GDI engine using a single-component fuel of iso-octane and a multi-component fuel containing equal initial mass fractions of iso-pentane, iso-octane and n-decane to represent the light, medium and heavy fractions respectively. A detailed description is provided of the impingement, Leidenfrost temperature and liquid film modelling approaches.

Two impingement processes are evaluated in detail; plume 1 intake valve impingement and plume 6 cylinder liner impingement.

Knowledge of the fuel properties and wall surface temperature are used to predict the impingement regime for the different fuel fractions. Both the single-component and multi-component fuel were found to predominantly impinge within the free convection and nucleate boiling modelling regime of the impingement model, for both intake valve and cylinder liner impingement processes.

The Weber number and Laplace number have been evaluated for impinging droplets and found that whilst initial intake valve impingement occurred at 6°ASOI, it wasn’t until 10°ASOI that the Weber number of a significant number of impinging droplets had reduced sufficiently to move from a splashing regime and into a deposition regime with a liquid film being formed. Once a film is formed and the surface wetted, the liquid film
quickly grows due to the increased critical Weber number associated with a wetted surface.

Evidence of film stripping was seen from the lower surface of the valve head with ejected droplets having very large diameters (100-500μm) and slow velocities which could be a source of UHC and PM emissions later in the cycle.

It was also found that the influence of charge motion on plume distortion was predicted to be greater with the multi-component fuel. This caused the location of impingement and predominant film formation for the multi-component fuel on the top surface of the intake valve, whereas the single-component fuel impinged and primarily formed a liquid film on the side of the valve head. The onset from this is that the subsequent film stripping and child droplet formation processes would be different depending on whether a single- or multi-component fuel is modelled.

Cylinder liner impingement is also investigated. A qualitative comparison against experimental images indicated a good match with respect to the timing and positioning of impingement and subsequent deflection pattern of the droplets circumferentially around the liner wall. It is again found that evaluation of the Weber number and Laplace number of impinging droplets could be used to deduce the onset of liquid film formation.

Finally, the models used for determining the Nukiyama and Leidenfrost temperatures (used as critical inputs into the models for film boiling and impingement regimes), are predominantly based on experimental results for sessile droplets though it is known that they are a dynamic property also dependent on the droplet impingement characteristics. Yao and Cai\textsuperscript{42} proposed a model to determine the Leidenfrost temperature of impinging droplets as a function of the droplet impingement angle but the study had a number of limitations; the study was conducted with large water droplets, the model contained four user tuneable coefficients and was a function of saturation temperature rather than a corrected Leidenfrost temperature, as used in Habchi’s model. Further model developments in this area, including supporting experimental studies to investigate the sensitivity of Leidenfrost temperature to dynamic impingement conditions relevant to impinging sprays in engines, would further improve the accuracy of impingement and liquid film predictions in engines.
FUNDING

This work was supported by Jaguar Land Rover and the UK-EPSRC grant EP/K014102/1 as part of the jointly funded Programme for Simulation Innovation.

REFERENCES


