ISSN: 1231-4005
e-ISSN: 2354-0133
DOI: 10.5604/01.3001.0012.4800

COMPUTATIONAL MODELLING OF THE FUEL INJECTION AND COMBUSTION IN A DIESEL K6 ROTARY ENGINE

Max Mason, Miroslaw L. Wyszyński

The University of Birmingham
Department of Mechanical Engineering
Edgbaston, Birmingham B15 2TT, United Kingdom
tel.: +44 7968 157 909
email: m.l.wyszynski@bham.ac.uk

Owen Jordan, David Gibson

Railfast Intermodal Limited
326 Woodway Lane, Walsgrave
Coventry, CV2 2LF, United Kingdom

Abstract

This paper outlines the methods and results of computations completed using the ANSYS Fluent code modelling the fuel injection and combustion within the K6 engine, a new form of rotary engine in which the fuel is injected in an arc across the top of the cylinder. The model uses the DPM Model in conjunction with a dynamic mesh and non-premixed combustion models to treat the injection as liquid diesel evaporating to $C_{12}H_{23}$. The outcomes of this model are presented in images displaying the distribution of temperature, and fuel and CO_2 concentrations. The limitations pertaining to the maximum injection angles are also studied. The simulation is found to be effective and the results suggestive of successful, clean and complete combustion while presenting some matters, which require further investigation. The article presents temperature within the combustion chamber at various crank angle degrees,) velocity of fluid within the combustion chamber, effects of impingement with injector offset on temperature and fuel concentration, fuel concentration demonstrating impingement, in cylinder temperature curve

Keywords: *combustion engine, combustion process, combustion factors, fuel injection, ANSYS Fluent code*

Nomenclature

Species Transport – the calculated conversion of one substance (species) to another involving heat change, chemical change and resulting fluid properties change,
Impingement – the (normally) derogatory effect of the proximity of a solid surface on a combustion reaction due to restriction of air flow or turbulence effects,
PDF – a probability density function used to take into account chemistry and turbulence variation when modelling the species transport involving mixtures [16],
Wetting – the derogatory effect of inferior liquid fuel combustion on a solid surface,
NOx – general term for nitrogen oxide pollutants,
TDC – Top Dead Centre, the highest position of the piston within a cylinder,
BDC – Bottom Dead Centre, the lowest position of the piston within a cylinder,
BTDC – Before Top Dead Centre,
ATDC – After Top Dead Centre.

1. Introduction

The combustion of fuel to release energy is the working concept of all internal combustion engines, it therefore follows that the quality of the combustion within the cylinder is an important factor in the success and efficiency of any engine and all efforts must be made to observe and improve the combustion pattern in an engine as part of its development.

This report aims to model and discuss the fuel injection and combustion in a K6 engine. This will be achieved through producing a literature review pertaining to the combustion within a cylinder, producing a working model of a K6 engine for 30º BTDC to 30º ATDC within Fluent and using this model to review the success of the combustion within a K6 cylinder

The K6 engine is a novel design of rotary engine developed by Owen Jordan in conjunction with the University of Birmingham. The general engine arrangement consists of a fixed stator inside of which a piston arrangement, of variable number of pistons, rotates around the crankshaft (see Fig. 1). The pistons and cylinders rotate around non-aligned centres leading to the reciprocating motion of the pistons, whose motions relative to the cylinder are non-sinusoidal. Working on a conventional two-stroke engine cycle the K6 has a pattern of ports on the stator which open and close as the cylinder passes them and the design maintains a degree of scavenging as the fresh charge of air expels the old exhaust gases. The design is similar to and improves on the Gnome engine [1] although the complex moving injector system is replaced by a single injector on the stator. The aim of the engine is to provide a higher power to weight ratio and reduced imbalances while utilising a simpler design. The design is patent pending [2].

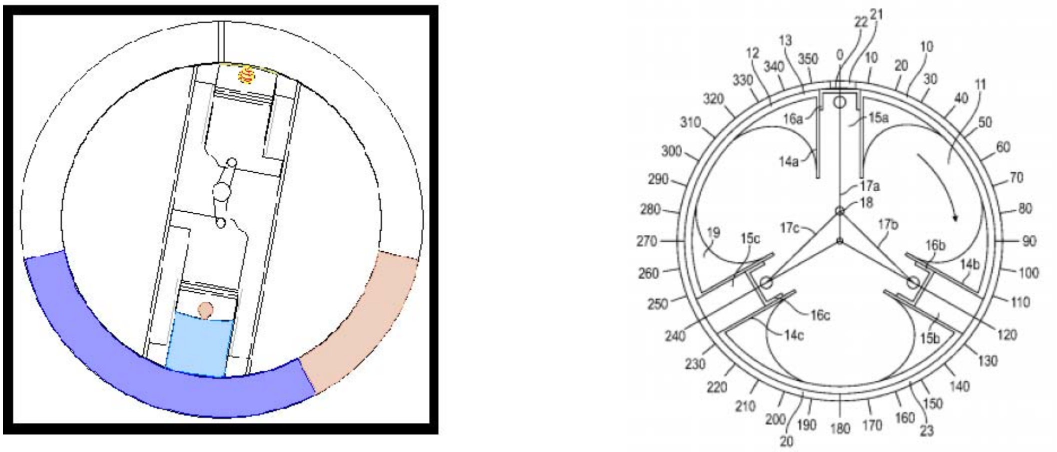


Fig. 1. Views of the K6 Engine as Found in: A. Explanatory Material and B. GB Patent No. WO2017168128A1

Due to the unusual piston configuration and motion within a K6 engine, at the point of injection there is a large horizontal translation when taking the cylinder as a reference frame. This leads to the fuel being injected in an arc across the top of the combustion chamber rather than at a single point as in a conventional DI engine. Combined with the effects of unique piston crown design and injection constraints this engine presents a largely changed combustion pattern, which this article aims to investigate. It is hoped that the alternative injection and combustion will aid efficiency and power from the engine adding to the advantages of the K6 engine over its competition.

2. Combustion Factors

There are many factors, which effect the success of the combustion of diesel within a cylinder, and several parameters, which are used to quantify and measure the quality of combustion. It is important to gain a background of these properties and previous work on the topic before investigating the K6 engine.

While many of the operating conditions in the K6 engine are unchanged from a conventional 2-stroke the major difference effecting the combustion is the bulk mixing of the fuel with the air. Smith and Timoney [17] separate the major contributing factors in fuel-air mixing into three categories:

– the fuel injection process,
– air motion effects, and
– combustion chamber geometry.

Through looking at these three sections, components can be considered individually, though perhaps are not fully understood [17]. Each of these is influenced by several factors, none more so than the fuel injection process.

In the 1988 SAE technical paper, entitled "Effects of Fuel Injection on Diesel Combustion" [3] Beck et al discuss the "fuel spray effect", the phenomena by which the spray consists of a liquid portion and a droplet portion. As the fuel combusts where it is in direct contact with air the far greater surface area of the droplets means combustion is far more successful with a large droplet region. The breakup of the liquid fuel into microscopic droplets (termed atomization) has been studied in great depth in papers such as "Investigation of fuel spray atomization in a DI heavy-duty diesel engine and comparison of various spray breakup models" [4] published in "Fuel". Although this article focuses on the CAE modelling of fuel breakup it touches on the importance of achieving "proper spatial and temporal distribution of the fuel spray". As the K6 injection is spread over, much of the cylinder the distribution is likely to be superior to that of a standard DI engine although the fuel breakup may behave unusually.

Within the K6 cylinder, the effect of the translating injector will not only affect the mixing of the fuel with air but could also lead to impingement on the side of the combustion chamber at the extremes of the injection period. Although this exact problem is unique to the K6 engine, the effects of impingement in general are well understood due to work such as "Effect of flat-wall impingement on diesel spray combustion" [5] from the Proceedings of the Institution of Mechanical Engineers.

The standard view on impingement as expressed by [5] in "Effects of spray impingement, injection parameters, and EGR on the combustion and emission characteristics of a PCCI diesel engine" [6] is of a purely derogatory impact, however Li et al find that at an appropriate impinging distance combustion is enhanced. This effect may be seen in the K6 engine as the distance is constantly changing although it is expected the majority of impingement is problematic.

In order to prevent impingement, the injection pressure must be closely monitored as "flat-wall impingement causes diesel combustion to deteriorate when the liquid phase–wall interaction occurs" [5]. However, as stated in "Effects of Injection Pressure and Nozzle Geometry on D.I. Diesel Emissions and Performance" [7] higher injection pressure leads to improved atomization and fuel-air mixing, although a trade-off is to be made with increased NOx production.

3. Methods

3.1. ANSYS Fluent

All of the computational fluid dynamics (CFD) for this project was completed in the ANSYS Workbench system with calculations done using the Fluent code. Fluent was chosen for its high adaptability, with several models preprogramed for each process and with user defined functions (UDFs) inputs available it can be used for a large range of simulations. It is also a highly user-friendly package.

To analyse the combustion over time the Fluent model is produced as a time transient simulation. This calculates results at several discrete time-steps of predetermined size with the result of each being computed from the output of the previous iteration. This not only allows the user to model systems, which change over time but view data for each time-step, giving a more useful insight into the solution. A drawback of this method is the greatly increased computation time, as each time-step must be calculated individually. The time step size used in this project is 0.0001 seconds, approximately 1 crank angle degree.

3.2. Engine Conditions

Before any computational model can be created, it is first necessary to find the conditions present within the cylinder. Several parameters must be included in the programming of Fluent, some of which are known through design however a number are required to be calculated.

Tab. 1. Engine Design Specifications as Modelled

Engine Speed (rpm)	1600
Compression Ratio	17
Stroke (m)	0.08
Bore (m)	0.1
Crank Stroke Ratio	0.55
Injection Start (DBTDC)	5
Injection Duration (CA)	17

Based on the data in Tab. 1 the quantity of fuel required can be approximated by assuming stoichiometric combustion, standard diesel fuel, and a volumetric efficiency of 100%. Injection timings are also calculated with this information and can be found in Tab. 2 below.

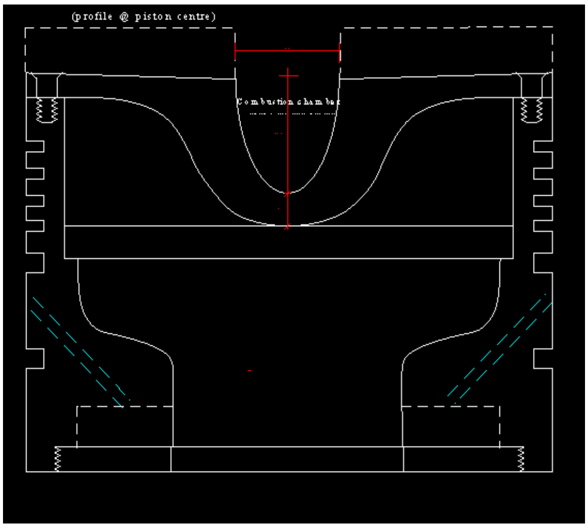
Tab. 2. Injection Parameters (all times after bottom dead centre)

Cycle Time (s)	0.375
Injection Start (s)	0.01786
Injection Duration (s)	0.00177
Mass of air in cylinder (g)	1.0187
Stoichiometric Mass of Fuel (kg)	0.0679
Fuel Injection Rate (kg/s)	0.03835

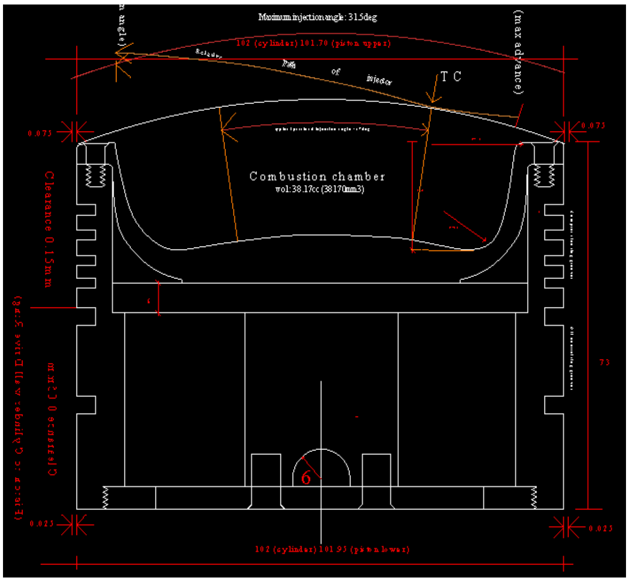
3.3. Geometry

Due to the unusual path of the injector relative to piston, an unorthodox piston crown has been designed to include a long, deep combustion chamber to allow the fuel to be injected along the length of the whole combustion chamber without impinging on the top of the piston (see Fig. 2). To maintain the correct depth along the length of the combustion chamber while conserving the required total compressed volume, the chamber was made very narrow. This leads to a reduction in surface area on the base of the combustion chamber. This is an untested new design which may have an unexpected or derogatory effect on the air motion within the cylinder, it is therefore important that when testing the combustion within the K6 engine it is done including this piston crown design and not in isolation.

Based on technical drawings of the prototype engine in Fig. 2a and 2b, a 3D CAD (computer-aided design) model was produced in Solidworks™. The geometry input into Fluent is a 3D model of the fluid body rather than of the cylinder itself, thus it encompasses the shape of the piston crown at the bottom, the cylinder walls at the side and the curve of the stator at the top (see Fig. 2c and 2d). All the simulations take the cylinder as a reference frame such that the only motion modelled is the piston and the apparent rotation of the stator.



a) Section view of piston head across gudgeon pin



b) Section view of piston crown through gudgeon pin

c) Front view of CAD model of combustion chamber at BDC,

d) Isometric View

Fig. 2. Engineering drawings and views of finished CAD model of Combustion chamber

3.4. Simulation

3.4.1. Meshing

The solver does not take the geometry as a solid body but rather a mesh of interconnected nodes at which the solution is calculated for each time-step. As the computations are done for each node an increase in element number leads to a diminishing returns improvement in accuracy and precision with a trade off with calculation time.

To model the bulk motion of the air in the cylinder and introduce turbulence to the fluid a dynamic mesh was implemented to simulate the movement of the piston rising and falling. The use of a dynamic mesh also allows for accurate determination of impingement on the piston crown as the distances change in real time. The dynamic mesh works by producing a new mesh each time-step, with the same nodes shifted to allow for the change in geometry. Using a velocity profile produced in notepad based upon the true piston velocity as calculated, the section of the mesh denoted as the piston crown is moved each iteration. The K6's very short conrod: stroke ratio means it has a non-standard reciprocating velocity, which was calculated previously. The thin element piston crown is treated as a rigid body, which moves only vertically whereas the 3D cylinder above is compressed and is re-meshed. This leads to an increase in the mesh density,

especially towards the end of the movement when the compression ratio reaches 17:1 (the compression ratio of the cylinder). The increase in mesh quality over time means a lesser quality mesh is required to be produced initially. Indeed, any mesh with too many nodes leads to the creation of negative volume elements in the remeshing process, causing an error in the solver, as the required edge length becomes smaller than the minimum value.
A mesh was created from the Solidworks body using the ANSYS meshing package with the parameters as shown in Tab. 3.

Tab. 3. Mesh Properties

Node Number	2487
Element Number	11565
Skew	0.54
Element Size	Average Approx. 5e-8 m^3

3.4.2. Models

For each process within the cylinder, a model needs to be either selected for the preprogramed list or entered manually.

The turbulence model used was the Realizable K-epsilon model, a two-equation model described in the ANSYS customer training material as accurate for both planar and round jets and superior for flows involving rotation [8]. This model was left mostly as default as other models are used for other aspects of the investigation.

As discussed by Hossainpour et al [4] there are several different models for the breakup of droplets within the injection stream. In the article several are tested, and each is found to have its own benefits but for this study the Wave model was chosen as it is available in Fluent and is recommended for high-speed fuel injection [9]

For the calculation of combustion, the species model within Fluent is enabled using the non-premixed combustion sub model with non-adiabatic energy treatment. Using this model requires the fuel to be described as an evaporating liquid, the vapour of which then combusts with the oxidant as predetermined using a probability density function (PDF) table [16]. While the liquid is modelled as liquid diesel with all the properties accurate, once evaporated the vapour is taken to be $C_{12}H_{23}$, the average length carbon chain in diesel [10], to simplify the calculations.

3.4.3. Injections

The DPM (Discrete Particle Model) was used to inject and track individual particles to simulate the fuel injection. Rather than a single moving injector, several stationary injections were used with one for each time-step of the computation along the true path of the injector. As the solution is calculated, only at each time-step and not between, there is no loss of information using this technique.

The true injection path is an arch and the injection is always angled towards the centre of the stator. The positions and trajectories of each injection were calculated based on the angle of the cylinder within the stator at the time of each injection.

The injection was of the droplet type using a simple point cone of angle 9^{o} [11] injecting liquid diesel at 300K.

3.4.4. Simplifications and Assumptions

Due to the limited time and resources, some simplifications and assumptions have been made to produce the best *working* model possible within the practical constraints. While the true engine cycle includes the gaseous transfer at the ports this model was only designed to investigate the fuel injection and combustion. The model therefore does include the lasting effects of the inlet and outlet efficiencies and imperfections. Also, the real engine works on a cycle and exhaust gases (especially in a two-stroke engine) will never be fully evacuated while this model presumes the fresh charge to be pure air.

As the model is only able to run the processes included in the fuel combustion systems, the decision was made to start the computation only at the beginning of the injection cycle. Any changes prior to this can be included within the initial conditions and found in separate calculations. To this end, a simple simulation was run to find the initial temperature, pressure and velocity distributions within the cylinder up until 5° BTDC.

The injection produces a solid cone, which is akin to that of a single-hole injector, the type most likely used for this engine as a spray angle wider than 15° would lead to wetting of the sides of the combustion chamber due to its narrow geometry. This type of injection does have problems as the narrower angle means bulk mixing is lessened, in the future simulations could be run to determine the ideal angle for this geometry.

All the engine parameters have been taken as they were found in the source material and no work has been done to improve the performance of the engine. Future work will need to be completed to optimise each variable to achieve maximum performance.

4. Results

A drawback of this type of modelling as compared to 1-dimensional engine modelling is the lack of quantitative data as the results are primarily distributions of various factors displayed as images. However, by analysing these visuals it is possible to gain insights into an individual parameters effect on engine performance simply not possible through numerical analysis alone.

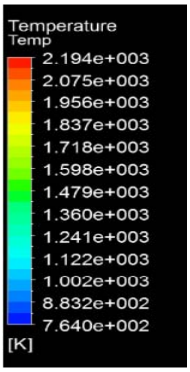
The following figures display the temperature within the combustion chamber at varying crank angles from which many important conclusions can be drawn on both the success of the combustion and success of the model.

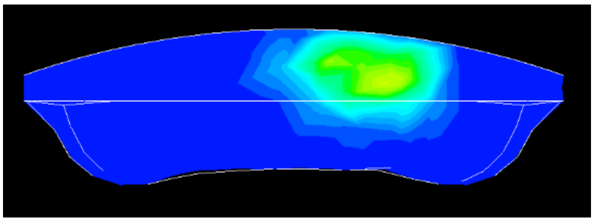
It is clear to see from Fig. 3 that the injection starts in the top right of the image and moves across to the left, representing a clockwise rotation of the cylinder within the stator.

An immediate point of interest is the lack of ignition delay. Defined as the time interval between the start of injection and the start of combustion, in normal engines a period of around 0.5 ms [12]. However, as seen in Fig. 3a the combustion appears to start instantaneously, in sync with the beginning of the injection (5°BTDC). Ignition delay is strongly correlated with the temperature and pressure within the cylinder at the time of injection, with higher values leading to a shorter delay. The reduced cooling experienced by combustion gases due to the incoming fuel may start to explain the over-fast combustion but there are some chemical processes which limit how quickly combustion can begin, and the delay is limited to at least 0.4 ms which is absent from the results. There is an inbuilt model for ignition delay within Fluent, but it is not available for non-premixed combustion as this is which may explain the lack of delay.

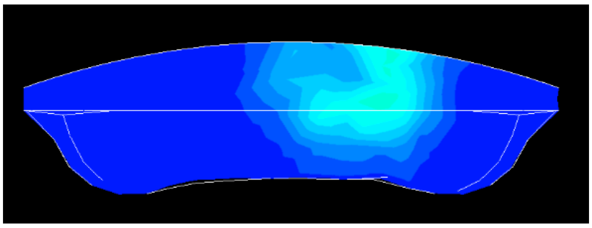
After this initial burst of high temperature at the injection point the working fluid cools again to almost normal levels, suggesting that the source of heat has changed, and ignition as expected occurs around 1 degCA BTDC as shown by the higher temperature yellow region in Fig. 3d. This gives an ignition delay of 4 degCA, a very fast but possible and improved value over a standard DI engine.

As witnessed in Fig. 3 and in keeping with standard fuel injection, a roughly spherical region of fuel droplets is formed where the fuel is injected with combustion taking place primarily at the boundaries of this region. This can clearly be seen in Fig. 4, which displays the fuel concentration in comparison to the temperature and CO_2 concentration at 1 degCA ATDC.

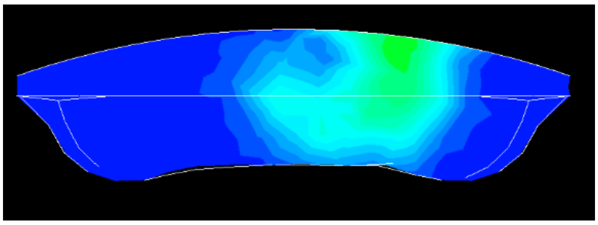




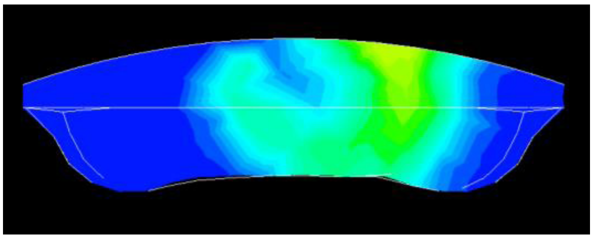
a) In-cylinder temperature at 5 degCA BTDC



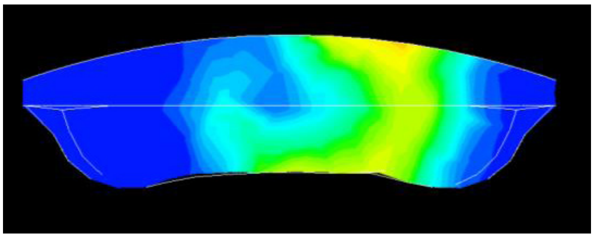
b) In-cylinder temperature at 3 degCA BTDC



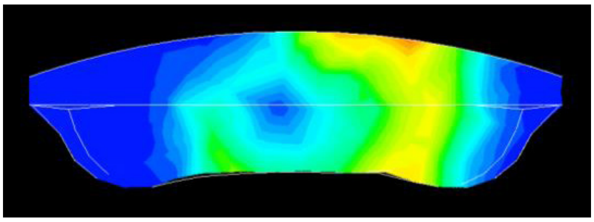
c) 1 degCA BTDC (time-step 25)



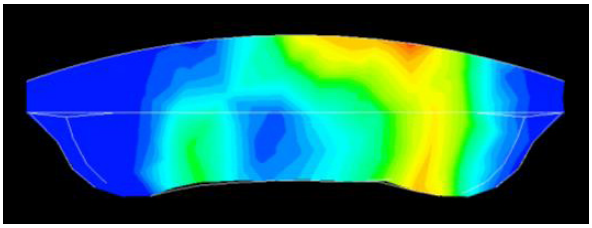
d) 1 degCA ATDC (time-step 27)



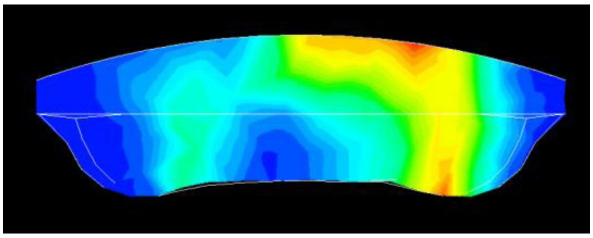
e) 3 degCA ATDC (time-step 29)



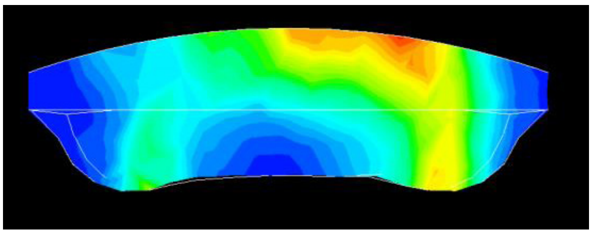
f) 5 degCA ATDC (time-step 31)



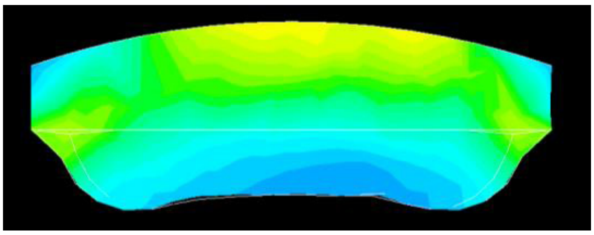
g) 7 degCA ATDC (time-step 33)



h) 9 degCA ATDC (time-step 35)

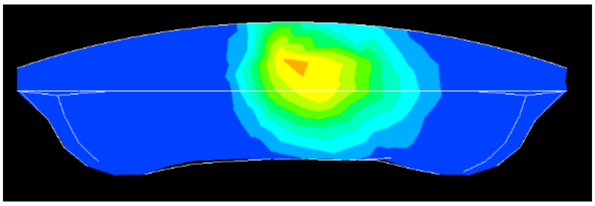


i) 12 degCA ATDC (time-step 38)

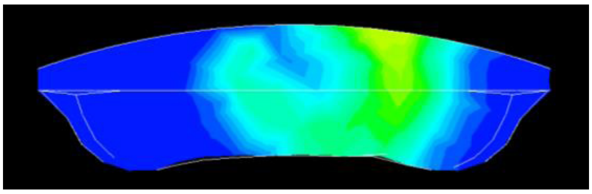


j) 16 degCA ATDC (time-step 42)

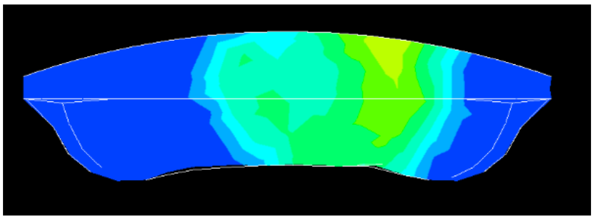
Fig. 3. Temperature (K) within the K6 Combustion Chamber at various Crank Angle Degrees



a) Fuel Mass Concentration at 1º ATDC



b) Temperature at 1º ATDC



c) CO2 Concentration at 1º ATDC

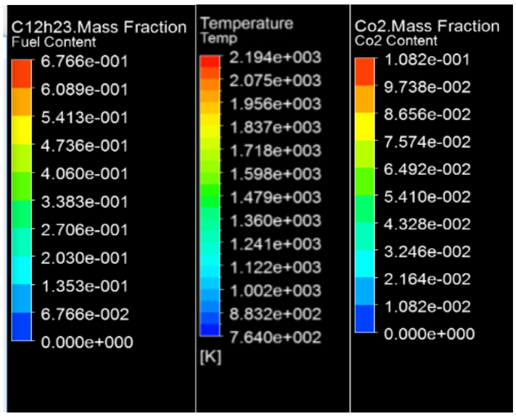


Fig. 4. Conditions within K6 Cylinder at 1º ATDC

Although there remains a clear circular region where combustion is taking place there are some differences with standard combustion. As the injection moves to the left it leaves behind the existing bulk of fuel, extending it to the left while getting further from the initial combustion front. This means fuel enters in a very different atmosphere depending on which side of the fuel bulk it is situated. On the left side of the fuel mass, the droplets are in a region of high O_2 content (Shown in Fig. 4c by the lack of CO_2) but the temperature is too low for spontaneous combustion as it is cooled by the incoming fuel. On the right side however, the droplets are in a very hot region with an atmosphere high in CO_2 leading to a richer type combustion. These conditions are exasperated later into the injection with the right side getting hotter and more carbon heavy while the left side is constantly injected into a fresh charge of air.

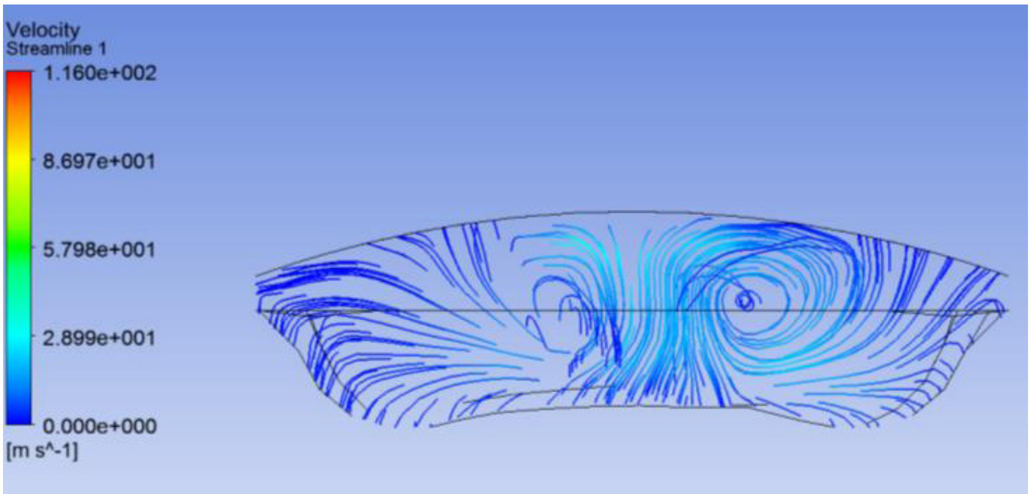
These unbalanced conditions are arguably an improvement on standard fuel injection as the normal conditions at the spray border are closer to that of the right-hand side of the K6 combustion but have increased fuel cooling. However, with the right hand side receding evermore into the exhaust gases it is possible that combustion that is more incomplete will be found in this region leading to the production of unwanted exhaust gases.

Throughout the duration of the injection, the combustion front does expand and traverse the exterior of the spherical region of fuel from the point of ignition. Initially most of this movement is downwards as the flame front spreads underneath the fuel bulk and reaches the left-hand side as visible in Fig. 3f and 3g. This means some fuel does combust on the extreme left of the fuel-rich central section in very lean environments soon after ignition; however, the cooling effect of the fuel coming into that region means this is kept at a minimum.

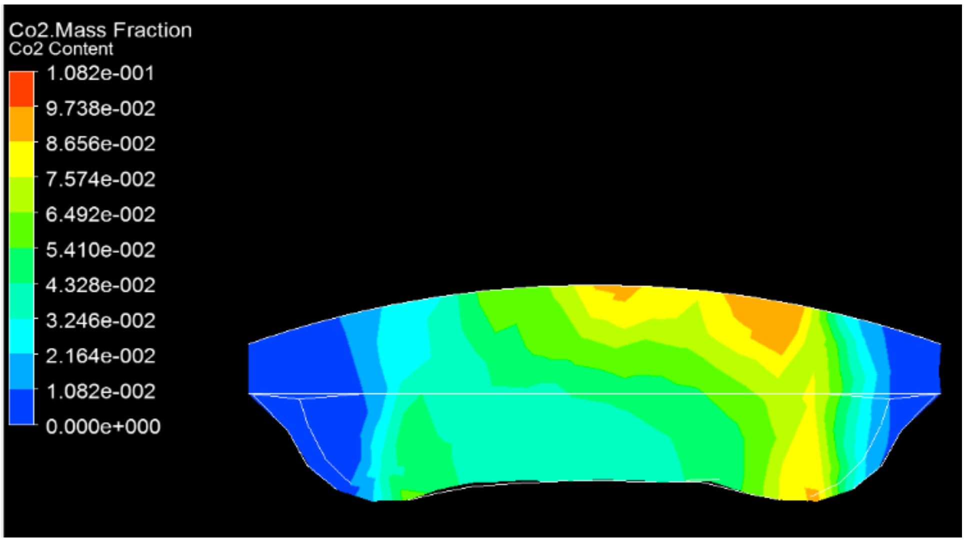
At around 6º ATDC the fuel mass is low enough such as to prevent the flame front from travelling along the bottom but at this point, the injection is at a lower angle and further to the left, which allows the hot region to very quickly spread across the top. It can be seen from Fig. 3i and 3j that as the injection ends, the area that the injection previously cooled and kept combustion free on the left side is very quickly heated to the point of combustion. By 15°ATDC the entire combustion chamber can get hot enough for combustion.

This combustion pattern is unusual but by no means problematic. While the combustion is confined at first to the right-hand side, it does eventually reach the left-hand side and all the fuel. It also suggests a steadier burn rather than an initial delay followed by a sudden and large heat rise. This coupled with the quickly descending piston means the temperature (and therefore pressure) rise will be reduced while still supplying the same energy to the piston.

As shown in Fig. 5a, most of the air movement after ignition is separated into eddies on either side of the combustion chamber. This is good for localised mixing but means fresh air is not brought from the left side to the right during combustion. There also exists a problem with the location of fuel and unused air at the end of the combustion, the fuel is mostly on the left but there remains fresh air on the right trapped behind a region of exhaust gases (see Fig. 5b).



a) Velocity of Fluid within the K6 Combustion Chamber at 3° BTDC



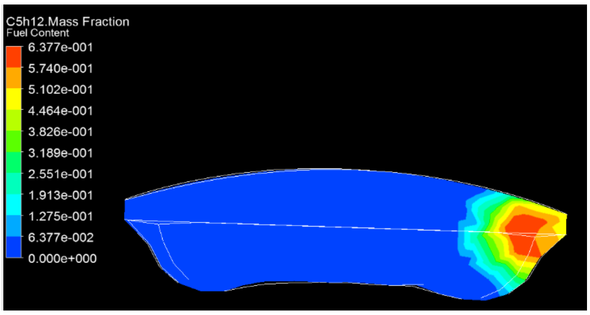
b) CO2 Concentrations within K6 Combustion Chamber at 12°ATDC

Fig. 5. Problems with bulk mixing in late stages of combustion within the K6 cylinder

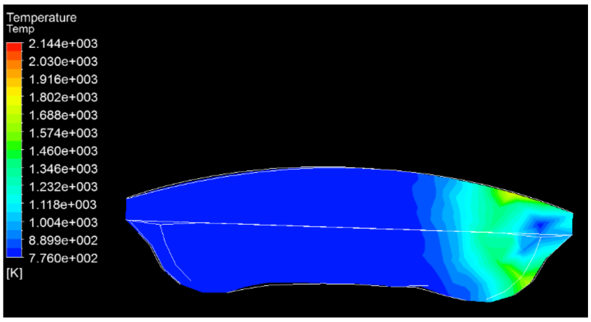
Due in part to this problem coupled with a stoichiometric amount of fuel this model predicts the fuel will be completely combusted only at a crank angle of 82° ATDC, much later than the Ricardo Wave prediction of around 60° ATDC [13]. A slower burn is not fundamentally a problem, as slower combustion leads to lower temperatures and therefore reduced NOx production. However, if fuel is burnt in an oxygen lean atmosphere, incomplete combustion will produce soot and carbon monoxide, which may lead to an engine not passing the strict pollutant regulations [14].

One solution may be to offset the injector from TDC. This leads to the injection starting further to the right and the piston reaching TDC before the injector reaches the centre of the cylinder. This does not remove the problem of an area of low O2 towards the right of the cylinder but reduces the amount of unused air behind the area, effectively moving the current combustion pattern to the right.

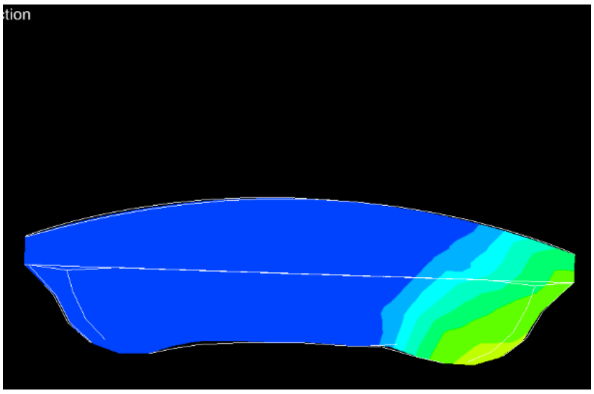
The problem with this solution is the increased likelihood of impinging on the sides and bottom of the combustion chamber at the extreme right of the cylinder as shown in Fig. 6. With an offset of 12°, the fuel spray hits the wall and burns very slowly and incompletely. With any offset of more than 8°, the right-hand side of the fuel mass is not exposed to air and therefore does not burn. This is shown in Fig. 6d where there is a band of high temperature on the left side of the fuel but not on the right. This same principle can be applied to the reverse side where any extremely late injection timing would result in impingement on the left side of the cylinder.



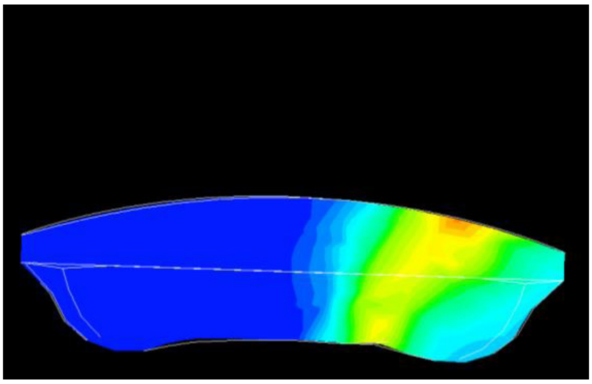
a) Fuel Concentration as Effected by Impingement at 4° BTDC



b) Temperature as Affected by Impingement at 4° BTDC



c) Fuel Concentration as Affected by Impingement at 1° ATDC



d) Temperature as *Affected by Impingement at 1° ATDC*

Fig. 6. Effects of Impingement with Injector Offset by 12° on Temperature and Fuel Concentration

Impingement is not only a problem at the left and right extremes of the cylinder. As can be seen in Fig. 3g through 3j and 7, the temperature contours showing the edge of the fuel droplet mass appear only on the top and not the bottom. This is because there is no combustion-taking place on the lower boundary as the fuel is in too close proximity to the piston crown. As the fuel is tracked via the DPM when it is in the liquid phase, it is possible to show that the fuel has all evaporated before coming into contact with the solid boundary. There is, therefore, no wetting of the piston crown but nonetheless the immediacy of the piston is a problem.

This problem can be alleviated through the careful management of the injection pressure as a reduced pressure will reduce the injection velocity and the fuel will not reach the bottom of the combustion chamber before combusting.

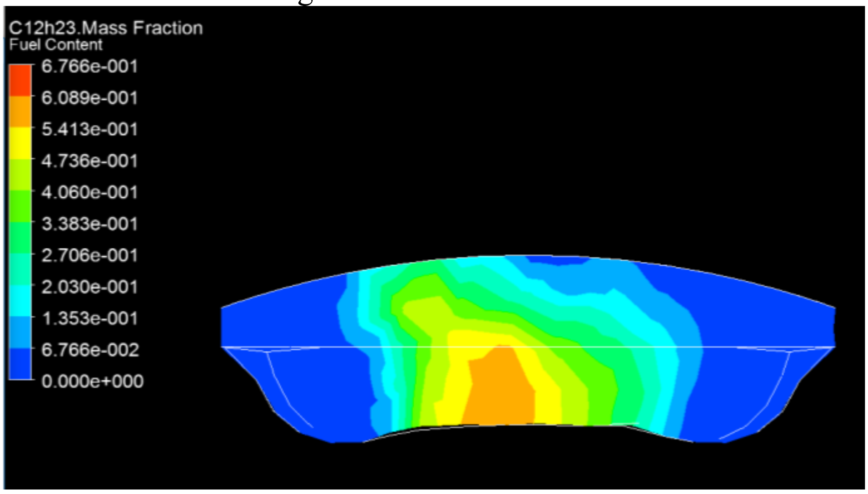


Fig. 7. Fuel Concentration Demonstrating Impingement at 9º ATDC

The maximum temperature is recorded as 2467K, a very similar value to that found by Yuekang Du [13] on Ricardo WAVE™ platform using 1D engine modelling, as shown in Fig. 8. This comparison acts as a validation to the relative accuracy of the results.

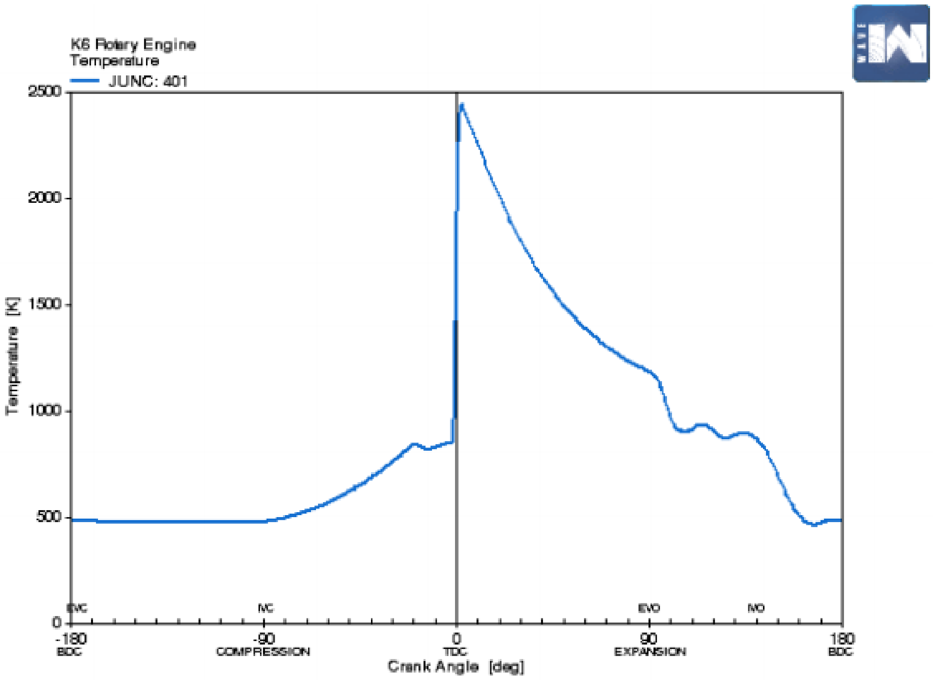


Fig. 8. In Cylinder Temperature Curve as Found in Ricardo WAVE™ modelling [13]

5. Conclusion

While every effort was made to produce a model of the highest quality, it is important to consider the scope of the study when building a simulation. They are time consuming and difficult to perfect, a great deal of research has been funded or carried out by automotive companies trying to advance their technologies beyond that of their rivals [15].

With this in mind, this study can be called a success. The model is hard to validate, as this engine has no real-world data with no prototype yet completed, but the agreement with other studies provides some confidence in the results. In theory it would be possible to apply this model to a known engine to achieve validation, but the changes could be large enough to take render it useless. The presence of the initial burst of heat remains an outlier and casts doubt on the quality of the model.

Beyond this the results are, within reason, as expected and go some way to justifying the production of a prototype and continued study. The combustion is complete and does not have any problematic signs:

– there are no areas of excessive heat,
– by the end of injection, the whole chamber has reached combustion temperatures,
– the temperature and therefore pressure rise is moderate and controlled and if the injection timing is matched to the descent of the piston it can improved further,
– all the fuel is combusted,
– there is no whetting of chamber walls.

There are some issues but overall, they can be solved through careful management of engine parameters:

– there is impingement on the bottom of the combustion chamber, but this can be reduced with the injection pressure,
– there is a region of unburnt air left behind by the injector, but this can be improved by adjustment of injector placement,
– this adjustment cannot be too extreme to avoid impingement or whetting at the ends of the combustion chamber,
– there is limited combustion in the left-hand side (see Fig. 3) of the combustion chamber.

There are some indicators that there may be advantages to this type of combustion over standard DI injection:

– ignition appears to happen very quickly due to the lack of cooling by incoming fuel, allowing near TDC injection timing,
– the right-hand flame front (see Fig. 3) is not cooled and therefore burns quickly,
– the fuel is spread throughout the cylinder and if injection durations and pressures are properly controlled it is feasible to expect very high-quality bulk mixing.

While this study suggests that the K6 engine will be able to function as expected, a great deal more work will need to be completed to adjust and improve each parameter and allow the K6 to achieve its full potential.

References

[1] Seguin, L., Lazare, A., *US Patent 959172,* United States of America 1910.

[2] Jordan, O. G., *Rotary internal combustion engine,* GB Patent application WO2017168128A1 GB, 2016.

[3] Beck, N., Uyehara, J., Otto, A., Johnson, W. P., *Effects of Fuel Injection on Diesel Combustion,* SAE Technical Paper Series 880299, 1988.

[4] Hossainpour, S., Binesh, A. R., *Investigation of fuel spray atomization in a DI heavy-duty diesel engine and comparison of various spray breakup models,* Fuel, pp. 799-805, 2009.

[5] Li, K., Nishida, K., Ogata, Y., Shi, B., *Effect of flat-wall impingement on diesel spray combustion,* 5, Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, Vol. 229, pp. 535-549, 2014.

[6] Kiplimo, R., Tomita, E., Kawahara, N., Yokobe, S., *Effects of spray impingement, injection parameters, and EGR on the combustion and emission characteristics of a PCCI diesel engine,* Applied Thermal Engineering, Vol. 37, pp. 165-175, 2012.

[7] Pierpont, D. A., Reitz, R. D., *Effects of Injection Pressure and Nozzle Geometry on D.I. Diesel Emissions and Performance,* SAE Technical Paper 950604, 1995.

[8] ANSYS, *Customer Training Material*, *L6,* 2010.

[9] ANSYS, *Fluent User's Manual, 22.7.2, Droplet Breakup Models*, 2006.

[10] Gupta, R. B., Demirbas, A., *Gasoline, Diesel, and Ethanol Biofuels from Grasses and Plants,* Cambridge, Cambridge University Press, p. 125, 2010.
[11] Prabhakara, R. G., Raju, V. R. K., Srinivasa, R. S., *Effect of Fuel Injection Pressure and Spray Cone Angle in DI Diesel Engines using CONVERGE CFD Code,* Procedia Engineering, Vol. 127, pp. 295-300, 2015.
[12] Lakshminarayanan, P. A., Aghav**,** Yoghesh, V., *Ignition Delay in a Diesel Engine*, *Modelling Diesel Combustion*, DOI 10.1007/978-90-481-3885-2, pp. 59-78, 2009.
[13] Du, Yuekang, *Modelling of a K3 Rotary Engine,* BEng Undergraduate Dissertation: University of Birmingham, 2017.
[14] Di Iorio, S., Mancaruso, E., Vaglieco, B. M., *Characterization of Soot Particles Produced in a Transparent Research CR DI Diesel Engine Operating with Conventional and Advanced Combustion Strategies,* Aerosol Science and Technology, Vol. 46, pp. 272-286, DOI: 10.1080/02786826.2011.620647, 2012.
[15] Wheelwrite, S. C., Clark, K. B., *Revolutionizing Product Development,* Simon and Schuster, ISBN 9781451676297, 1992.
[16] ANSYS, Fluent Handbook, pp. 14-1, 2001.
[17] Smith, W. J., Timoney, D. J., *On the Relative Roles of Fuel Spray Kinetic Energy and Engine Speed in Determining Mixing Rates in D.I. Diesel Engines,* Gas Turbines Power, pp. 212-217, 1997.

Manuscript received 17 July 2018; approved for printing 26 October 2018