Proceedings of Combustion Institute – Canadian Section
Spring Technical Meeting
McGill University, Montréal
May 15-18, 2017

CFD analysis of a Variable Compression Ratio HCCI Rotary Engine

Parth Panchal and Gaby Ciccarelli
Department of Mechanical and Materials Engineering
130 Stuart Street, Kingston, Ontario K7L3N6 Canada
Pouya Mottaghian and Roberto Fanara
Customachinery, Mississauga, Ontario

Abstract

This study investigates the operation of a new rotary engine using CFD. The rotation of the rotor and motion of a gate were successfully implemented using user defined functions and a dynamic mesh. Two-dimensional simulations were carried out using a single step reaction model and default Fluent liquid fuel spray and turbulence models. The simulations demonstrate that the unique engine design provides a mechanism for fuel air mixing and ignition.

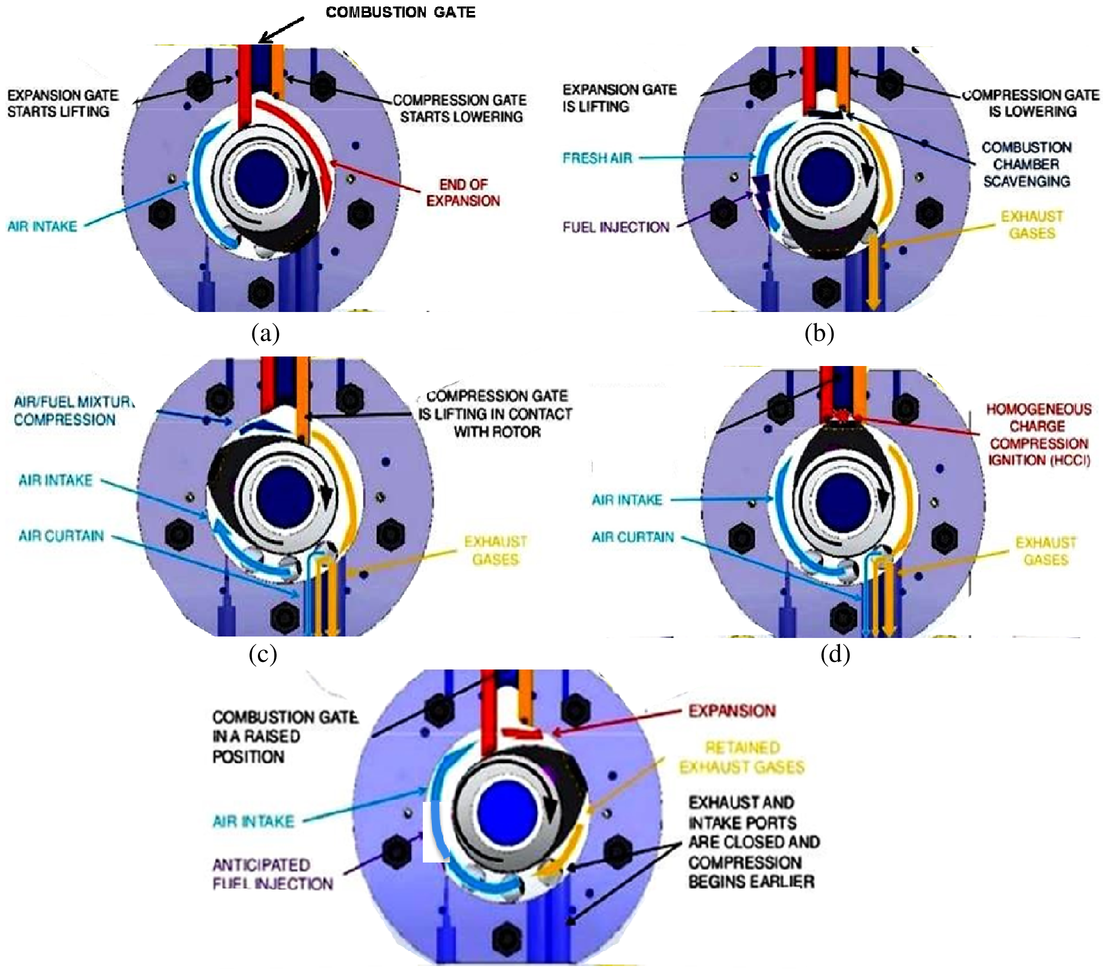
1. Introduction

In compression ignition engines, the soot-NOx paradox is an extremely challenging unresolved issue. HCCI is one of the most promising solution which combines the benefits of both SI and CI combustion modes [1]. Fuel lean premixed combustion helps to reduce or eliminate the formation of NOx and soot. HCCI engines have a high heat release rate at high load that results in knock. Rotary engines have many advantages, including high power density and the ability to run at high crank speeds because of inherent engine balance (there is no reciprocating masses). Power can be increased by stacking units side-by-side. The first rotary engine mass produced for automotive applications was the so-called "Wankel engine" [2], named after the inventor of the technology. Despite its many advantages, its ultimate downfall was poor thermal efficiency and emissions due to its high aspect ratio combustion cavity. A unique rotary engine based on the Homogeneous Charge Compression Ignition (HCCI) combustion mode patented by Customachinery Inc is under development at Queen's University. One of the unique features of this rotary engine is the ability to vary the compression ratio to control HCCI ignition timing. Unlike the Wankel engine it has a more traditional shape combustion cavity.

The rotary engine consists of a fixed stator and a rotating lobed rotor coupled to the output shaft. As shown in Fig. 1a, there are two gates at the top of the engine, i.e., compression gate and expansion gate. The intake port is located on one of the engine faces and the exhaust port is located on the stator. The compression ratio can be varied in real time by changing the vertical position of the combustion gate, located between the compression and expansion gates shown in Fig. 1a. A complete thermodynamic cycle requires one complete rotation of the rotor. Multiple processes occur simultaneously; the cycle will be described as a sequence where the inducted air is tracked. Shown in Fig. 1a the expansion gate is down and air has been inducted into the left side of the engine. In Fig. 1b the expansion gate retracts and the compression gate drops. During this scavenging process the amount of residual gas can be controlled, the maximum residual concentration corresponds to the case where the compression gate completed drops before the expansion gate starts to retract. Fuel injection can start during the scavenging process. The schematic in Fig. 1b shows a vertically oriented fuel injector on the bottom left side of the stator, the engine can also be operated in HCCI mode with fuel/air intake with no direct injector. The fuel-air mixture is compressed in Fig. 1c. When the rotor lobe approaches the 0^{o} CA position (top of the stator) both gates are retracted and autoignition occurs inside the combustion cavity defined by the position of the combustion gate, see Fig. 1d. Following ignition the expansion gate follows the rotor surface and the compression gate remains retracted so that the high pressure combustion products can expand on the right-hand-side of the engine, see Fig. 1e. The high pressure in this volume pushes the lobe producing rotor torque. After the rotor lobe passes the exhaust port, the combustion products blowdown and the cavity pressure drops to atmospheric pressure. The study includes an analysis of the flow and

*Professor, corresponding author, ciccarel@queensu.ca

combustion process involved using computational fluid dynamics, specifically to determine the optimum position of the fuel injector and fuel injection timing.



(e)

Figure 1. Schematic highlighting the following processes: a) air intake, b) fuel injection and combustion products scavenging, c) compression, d) ignition, e) expansion.

2. Methodology

ANSYS Fluent 17.0 was used to model compression, fuel injection, and autoignition in two-dimensions. The key component of the model is a dynamic mesh consisting of 298K elements that allows the modeling of the moving rotor and compression gate as shown in Fig. 2. User defined functions were used to prescribe the movement of the rotor and the compression gate. Unsteady pressure based SIMPLE solver is used where the Green-Gauss cell based option is selected for gradients. Pressure is solved by standard and PRESTO scheme whereas the other equations like conservation of species, momentum, energy and turbulent kinetic energy are solved by a first order upwind discretization scheme. The species transport equation and discrete phase model are solved simultaneously. Relaxation factors are used to help converge and stabilize the solution. The solution is advanced in time using an unsteady pressure based SIMPLE scheme with a time step of 5 µs.

The Fluent "discrete phase model" was used to model the spray in the form of droplets with the appropriate breakup models for the formed droplets. A Lagrangian-Eulerian multiphase formulation was utilized to simulate the interaction of the discrete and continuous phases. Unsteady particle tracking with spray sub models, such as pressure and temperature dependent boiling and TAB droplet breakup model, were used. The standard Fluent "Group injection" model was used, where fuel is injected from a point. Group injection with 10 droplet streams was used. The Rosin Rammler droplet distribution method was used with a minimum droplet diameter of 0.1 µm, a maximum diameter of 5 µm and mean diameter of 2.68 µm. The spray location, direction, and fuel mass flow rate are parameters that were investigated.

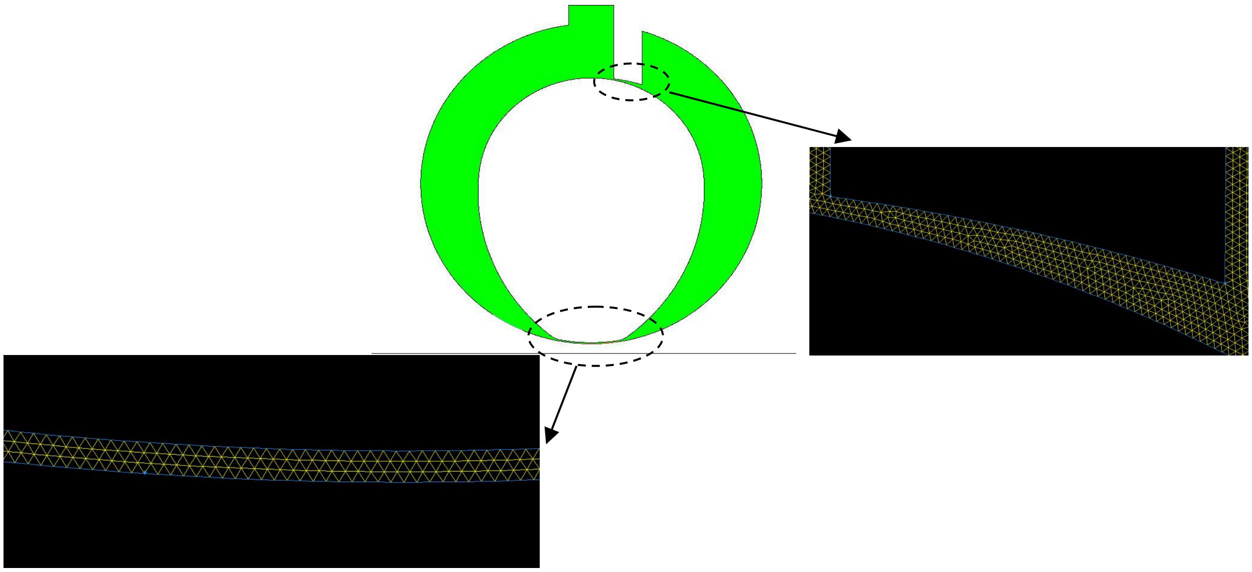


Figure 2. Schematic showing the two-dimensional geometry and dynamic meshing in the small gaps

The present study uses the Fluent "species model" which is able to model both premixed and non-premixed combustion. The single reaction mechanism was used with the default Fluent parametric values for heptane-air combustion. The Fluent "eddy dissipation concept" was used to model the turbulence chemistry interaction. In order to check the validity of the single reaction mechanism a constant volume 2D box, initialized with an elevated temperature of 800K, 900K and 1000K, was modeled. The pressure and temperature time-histories were compared to values predicted by the commercial chemical equilibrium code COSILAB using the reduced reaction mechanism from Wisconsin University [3]. The mechanism reproduces the small jump in temperature, before the main temperature rise, associated with the cool flame phenomenon. The Fluent simulation using the single reaction mechanism predicted the correct final pressure and temperature but is not capable of reproducing the cool flame temperature rise. The Fluent predicted ignition delay time was between 50 to 100 µs shorter than that predicted by COSILAB.

3. Results and Discussion

Calculations were carried out with a rotor and the stator diameter of 40 and 60 mm, and a rotor speed of 573 revolutions per minute. The gap between the rotor surface and the bottom of the compression gate is 0.25 mm, with 3 triangle mesh elements across the front gap (there are 9 elements across the rear gap), see Fig. 2. The gap between the rotor lobe and stator surfaces is also 0.25 mm , with 3 elements across the gap. Initial simulations were carried out with the fuel injector located on the left side of the rotor, as shown in Fig. 1. It was determined that with this injector location the fuel ended up coating the stator surface, which was deemed unacceptable. Subsequent simulations were carried out with the injector mounted in the combustion gate, results obtained with the injection directed off-vertical towards the expansion gate are shown in Fig. 3. The simulation starts with the rotor lobe positioned at 180º CA with the cavity between the rotor lobe and the expansion gate filled with air at 1 atmosphere and 350K. Heptane at 300K starts to flow at 20 ms with a mass flow rate of 5.52 gm/s and continues until the mass of fuel added results in a globally stoichiometric mixture, at simulation time 30 ms.

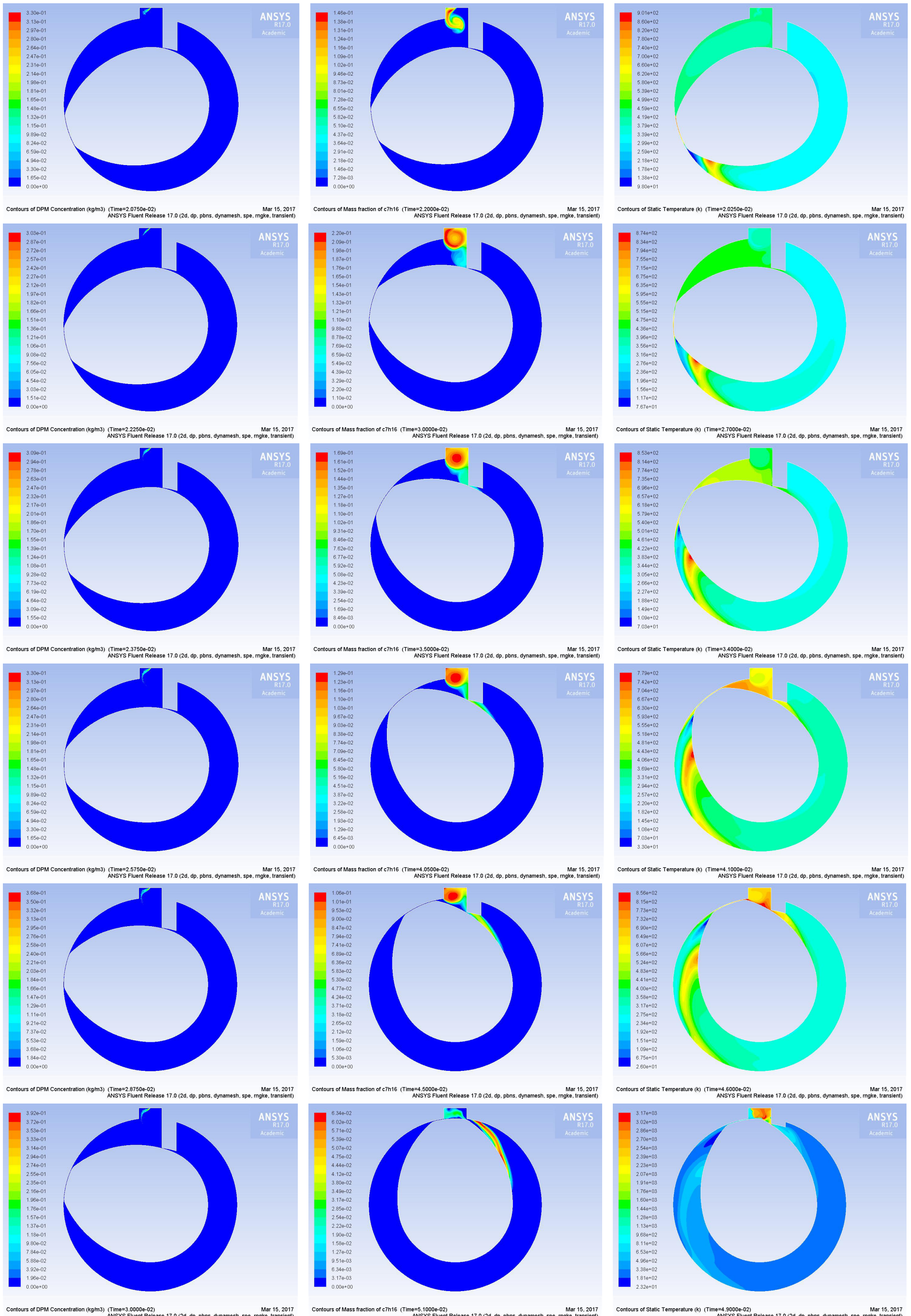
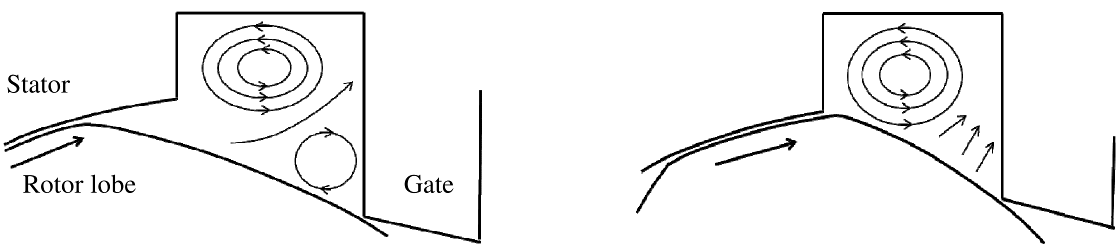


Figure 3. Sequence of contour plots for injection starting at 20 ms and stoichiometric fuel, a) liquid fuel concentration, b) gaseous fuel concentration, c) temperature. Time is shown in the caption below the images.

As the rotor rotates the lobe pushes the air in the annulus between the stator and rotor with a velocity on average equal to the rotor surface velocity of 2 m/s. This flow produces a counter-clockwise rotating recirculation zone in the combustion cavity (see schematic in Fig. 4) that is amplified to a velocity on the order of 30 m/s once the liquid fuel spray begins. At early times there are actually two counter rotating recirculation zones as shown in Fig. 4a, but as the rotor lobe approaches the combustion cavity only a single recirculation zone exists, see Fig 4b. The liquid fuel spray is observed in the contour plots in Fig. 3a. The length of the liquid spray remains roughly constant, as the fuel evaporates at the tip of the spray. The fuel vapour closely follows the air flow field in the combustion cavity that is characterized by the recirculation zone, refer to the contours in Fig. 3b. The highest concentration of fuel vapour is at the centre of the recirculation zone. There is evidence of fuel vapour diffusing towards the compression gate and leakage past the gap between the compression gate and the rotor surface. The contour plot at 51 ms shows the sudden disappearance of fuel vapour in the top right corner of the combustion cavity (blue region) associated with the start of combustion. The temperature contour plots are provided in Fig 3c. As the rotor lobe progresses the volume between the lobe and the compression gate decreases, resulting in an increase in the air temperature and pressure. Hot air leakage past the compression gate is observed, as well as past the gap between the moving rotor and the fixed stator. The evaporation of the fuel in the recirculation zone cools the air temperature locally, producing a temperature difference of up to 300K at 49 ms. The maximum temperature in the air before combustion starts is 856K that occurs at 41 ms. The following contour plot at 46 ms shows that the maximum temperature jumps to over 3000K, indicating the start of combustion. The point of ignition is just off the rotor surface and then combustion spreads inside the entire combustion cavity as a result of the strong recirculation zone.



(a) (b)

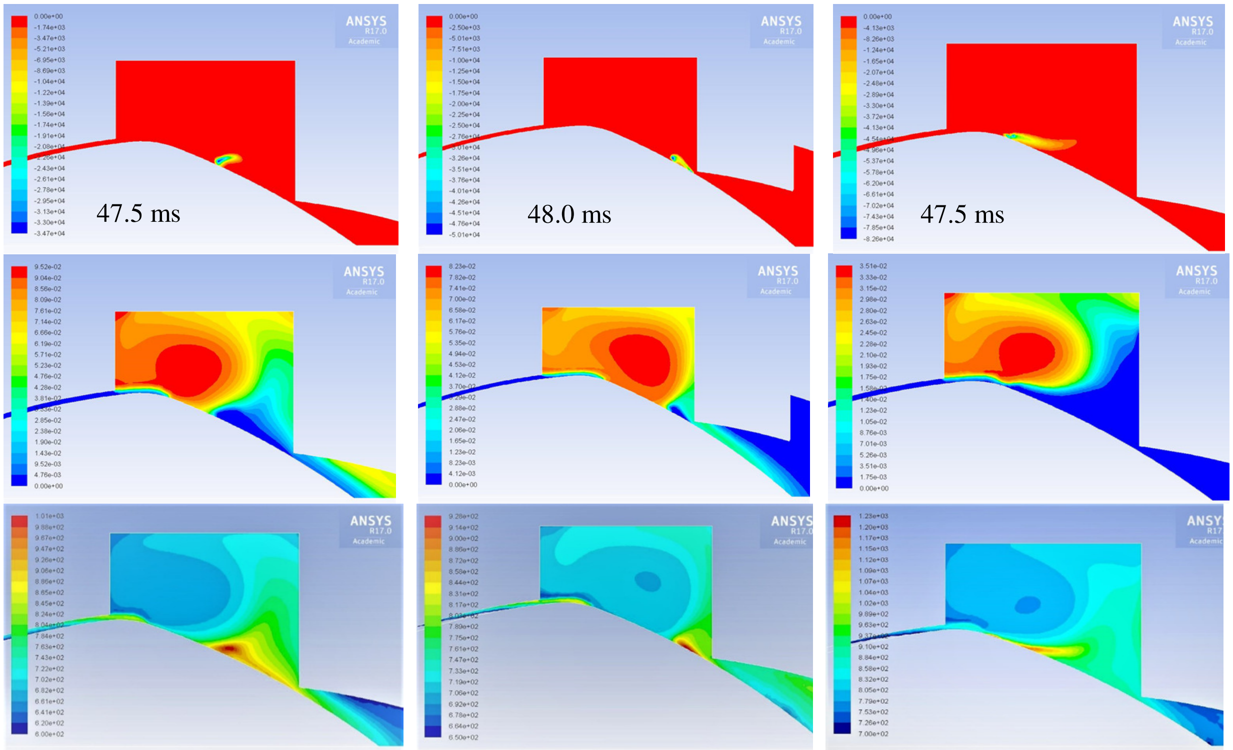
Figure 4. Schematics showing flow field at two different times

Similar results as those shown in Fig. 3 were also obtained with fuel injection starting at 30 ms and with 1/4 the fuel addition. The rate of reaction contours showing ignition, and fuel vapour concentration and temperature contours just before ignition are provided in Fig 5 for the three cases studied. In all three cases, the ignition time was very similar and occurred near the rotor surface, on the outer edge of the recirculation zone where the temperature is highest. For the later start of injection (SOI) of 30 ms the ignition point was closer to the compression gate, and for the ¼ stoichiometric case ignition occurred closer to the expansion gate. It was observed that the least amount of fuel leakage occurs for the 1/4 stoichiometric case (see Fig. 5c middle) as the fuel is contained largely inside the main recirculation zone at early times so that very little fuel makes it to the compression gate gap. There is more significant more significant

Future simulations will involve the study of the effect of compression ratio on the ignition time, as well as the effect of detailed kinetics on the ignition and combustion time.

4. Conclusions

The rotation of the rotor and motion of a gate were successfully implemented using user defined functions and a dynamic mesh using ANSYS Fluent. The fuel injection, mixing and ignition have successfully been modeled. The principle finding is that the recirculation zone in the combustion cavity promotes mixing and the generation of temperature and fuel concentration gradients that prevents uniform ignition inside the combustion cavity, a known issue with HCCI engines. It was found that the time for start of combustion is largely independent of the fuel equivalence ratio and start of injection time; however, these parameters do have a small effect on the relative location of ignition near the rotor surface.



(a) (b) (c)

Figure 5. Reaction rate (top), fuel vapour concentration (middle), and temperature (bottom) contours for a) stoichiometric with 20 ms SOI, b) stoichiometric with 30 ms SOI, c) ¼ stoichiometric with 20 ms SOI

Acknowledgements

Support for Parth Panchal was from the Ontario Center of Excellence TalentEdge Internship program

References

[1] A. Singh, A. Agarwal, Combustion characteristics of diesel HCCI engine: An experimental investigation using external mixture formation technique, Applied Energy 99 (2012) 116–125.
[2] Revolutionary Engine, Popular Mechanics. **113** (4): 96–97, 258. April 1960.
[3] A. Patel, S. Kong, R. and Reitz, Development and Validation of a Reduced Reaction Mechanism for HCCI Engine Simulations, SAE Technical Paper 2004-01-0558, 2004.