ABSTRACT
As part of the VECTIS code validation programme, CFD simulations were conducted to investigate direct-inject (DI) diesel engine combustion with the newly implemented Ricardo Two-Zone Flamelet (RTZF) combustion model. A small HSDI research engine was chosen as subject. The simulation programme consisted of 6 cases forming a complete injection time swing. CFD simulation results were compared directly against engine tests. Encouraging agreement was found for both the in cylinder pressure and NOx emissions.

INTRODUCTION
In recent years, due to increasing demand for fuel economy and tightening legislation for emissions, there has been growing requirement to develop more efficient and cleaner engines in a shorter time scale. Computational Fluid Dynamics, as a rapid and cost effective tool, is being increasingly used in different stages of engine design and optimisation. The combustion system is critical in engine performance and is therefore attracting great attention in CFD simulations. The challenge in simulating IC engine combustion using CFD is one mainly due to the complexity involving the interaction of flow, turbulence, spray and combustion. Further difficulty arises when simulating a DI engine where the combustion can take place under partially mixed conditions. The combustion model is required to be able to handle both premixed and non-premixed burning and their transition. Sufficient computational efficiency is also essential.

Over the years, the VECTIS engine CFD code developed by Ricardo Software has been widely used in the engine and vehicle industry and is favoured by design engineers due to its unique advantages in automatic mesh generation [1]. In early 1999, the Ricardo Two-Zone Flamelet (RTZF) combustion model was implemented. The RTZF model is a fractal-based combustion model which considers the possible finite pre-combustion air/fuel mixing and is therefore able to cope with both premixed and non-premixed burning conditions [2]. As a semi-empirical engineering model, it also has the advantage of high computational efficiency. The implementation of the RTZF model offers the potential for VECTIS to play a more important role in engine combustion system design and development.

Along with successful combustion applications reported by external users [3], Ricardo has been conducting a series of validation programmes internally to investigate the performance of VECTIS with the RTZF combustion model dealing with different types of engine combustion cases. The work presented here is part of this programme, focused on DI diesel engine combustion. The engine selected for this work was a small HSDI research engine which has been previously tested at Ricardo. The CFD simulation was conducted over a injection timing swing consisting of six cases. In the subsequent sections of this paper the engine configurations and technical details of the CFD simulations are described, and the comparison of VECTIS results with engine tests are discussed.

RTZF COMBUSTION MODEL
The RTZF combustion model is based on a ‘Two-Zone’ assumption whereby each computational cell is notionally divided into burned and unburned zones; the unburned zone is further divided into segregated and mixed regions (Fig.1). It is assumed that only in the mixed region that the air and fuel are mixed at the molecular level and ready for chemical reactions.

In non-premixed combustion, non-premixed fuel and air are located in the segregated region in the unburned zone, their mixing is continuously calculated. The newly mixed reactants are transferred into the mixed region where they are consumed by combustion. The combustion
products are finally passed into the burned zone. In purely premixed combustion, the calculation of mixing process is not required.

<table>
<thead>
<tr>
<th>Unburned Zone</th>
<th>Burned Zone</th>
</tr>
</thead>
<tbody>
<tr>
<td>Segregated region</td>
<td>Mix region</td>
</tr>
<tr>
<td>Air</td>
<td>Air</td>
</tr>
<tr>
<td>Fuel</td>
<td>Fuel</td>
</tr>
<tr>
<td>Residual Air</td>
<td>Residual Air</td>
</tr>
<tr>
<td>Residual Fuel</td>
<td>Residual Fuel</td>
</tr>
</tbody>
</table>

| Burned Air          | Burned Fuel            |

**Mixing border**  **Flame Front**

![Figure 1 Two-zone representation](image)

The general features of RTZF model can be briefly described as follows:

- The two-zone concept is introduced, enabling the local fuel/air mixing to be calculated when necessary. A unified model is therefore valid for both premixed and non-premixed combustion.

- The measured laminar burning velocity is directly used in the calculation. The turbulent burning velocity is evaluated from fractal geometry.

- The effect of flame quench is considered based on the basic flame dimensional analysis. The use of semi-empirical approaches provides a good accuracy in the burning rate calculation, together with a significant saving in computing time. The model is especially useful for engineering applications.

- The reaction chemistry is based on equilibrium, which can give quick and sensible results of combustion composition and flame temperature for a wide range of hydrocarbon fuels.

**ENGINE CONFIGURATIONS**

The general characteristics of the HSDI engine are given in Table 1.

![Table 1 Engine Characteristics](image)

<table>
<thead>
<tr>
<th>Name</th>
<th>HSDI</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type</td>
<td>1-cylinder</td>
</tr>
<tr>
<td>Bore</td>
<td>70</td>
</tr>
<tr>
<td>Stroke</td>
<td>78</td>
</tr>
<tr>
<td>Con-rod</td>
<td>124</td>
</tr>
<tr>
<td>Compression ratio</td>
<td>20.18</td>
</tr>
<tr>
<td>Swirling ratio</td>
<td>1.96</td>
</tr>
<tr>
<td>IVC</td>
<td>-158°ATDC</td>
</tr>
<tr>
<td>EVO</td>
<td>131°ATDC</td>
</tr>
</tbody>
</table>

**CFD ANALYSIS**

**Mesh Preparation**

Prior to the CFD simulation, computational meshes were generated using VECTIS. A complete surface model of the combustion chamber and piston crown was passed to VECTIS in STL file format, and the individual meshes were generated automatically. Local refinement was applied to all boundaries and within the piston bowl volume. The latter also covered the region near the injector when piston approaching TDC.

![CRANKANGLE = 268.00](image)

The movement of the piston/flow domain was resolved using the boundary motion feature of
VECTIS. During the solution, as the piston moves, the internal mesh structure deforms automatically to minimise the distortion of each individual cell. When the general cell distortion reaches a certain level the solution is re-zoned onto a new Cartesian mesh. In this study a total of nine meshes were used between IVC and TDC. Mesh sizes ranged from 64,000 to 28,000. Fig. 2 shows CFD surface geometry and computational mesh at 268° crank angle.

Cases Studied

CFD simulation was performed at 4500 rev/min and 10 bar BMEP. The fuel injection system was Lucas common rail with a centrally located 5-hole injector. The nominal start of injection (SOI) ranged from -18.5° to –8.5° ATDC as detailed in Table 2.

### Table 2. Fuelling and injection Timing

<table>
<thead>
<tr>
<th>Case No.</th>
<th>SOI (ATDC)</th>
<th>EOI (ATCD)</th>
<th>Fuelling mg/inj</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-18.5°</td>
<td>-2.5°</td>
<td>20.62</td>
</tr>
<tr>
<td>2</td>
<td>-16.5°</td>
<td>-0.5°</td>
<td>20.62</td>
</tr>
<tr>
<td>3*</td>
<td>-14.5°</td>
<td>1.5°</td>
<td>20.62</td>
</tr>
<tr>
<td>4</td>
<td>-12.5°</td>
<td>3.5°</td>
<td>20.62</td>
</tr>
<tr>
<td>5</td>
<td>-10.5°</td>
<td>5.5°</td>
<td>20.62</td>
</tr>
<tr>
<td>6</td>
<td>-8.5°</td>
<td>7.5°</td>
<td>20.62</td>
</tr>
</tbody>
</table>

* Baseline case

It should be noted that these timings represent the electronic pulse timings to the injector. Due to the electromagnetic and mechanical delay, it has been determined that the actual injection timings are about 10° crank angle later at this operating condition. This has been taken into account in setting up the spray timings.

An example of injection rate diagram used in the simulations is shown in Fig. 3. It was simplified from a measured curve.

**CFD Simulation**

Each full combustion simulation was completed in two steps; a compression stroke run from inlet valve closing (IVC) to start of injection (SOI), and subsequent combustion run from SOI to exhaust valve opening (EVO).

The measured in-cylinder pressure trace for the baseline case only was available at the time this simulation was conducted. As the result, a single compression stroke analysis was shared by all cases included in the injection time swing. Combustion runs were then conducted individually for each of the cases.

The compression run started at IVC, with initial velocities determined by solid body rotation about the cylinder axis at a swirl ratio derived from steady state measurements. The initial trapped mass was estimated based on the measured fuelling and global air/fuel ratio. Although there was no external EGR included in the cases studied, a small amount of internal EGR was always considered. 3-D unsteady CFD analysis was performed solving for momentum, continuity and energy. The k-ε turbulence model was used to describe turbulence transport, with the standard wall function approach for near wall treatment. Attention has been paid to match the simulated compression pressure to engine measurement, especially near SOI.

At the start of a combustion run, the spray and combustion models were activated. The spray modelling was based on the Lagrangian Discrete Droplet Method. The droplet break-up model used was that of Patterson and Reitz [4], with a droplet life time of 10. The Ricardo RTZF combustion model was used for the combustion modelling, using the default model coefficient $C_{\text{mix}}=20$. In order to predict NOx emission, the coefficients of Zeldovich NOx model were set up following the suggestion of Patterson [5], i.e. a NOx scaling factor of 2.5 was used in VECTIS simulation, and the predicted NOx mass fractions were then globally scaled up by a factor of 1.533 to convert predicted NO to NOx.
RESULTS AND DISCUSSION

Figure 4 presents the comparison of simulated and measured in-cylinder pressures for the baseline case (case 3). It was found that the general agreement between prediction and engine test was good. The ignition point is precisely captured, and the predicted in-cylinder pressure matches well in the early part of combustion. However, in the later stages of combustion a slight under prediction has been observed all the way through.

![Figure 4 Case 3: Predicted and measured in-cylinder pressures](image)

As we can see from Fig. 4, both measured and predicted pressure curves showed a change of gradient after the start of combustion. This clearly indicates a pre-mixed burn and quick flame development, most likely due to the effective fuel/air mixing produced by highly swirling flow in the small engine.

Figure 5 presents the in-cylinder spray, temperature and fuel mass fraction distributions from the CFD simulation of baseline case. The temperature and fuel mass fraction distributions are plotted for different crank angles on a vertical plane and a horizontal plane. At 355° and 357° crank angles, the vertical plane is located at a plane of spray. From 358° onwards, due to the strong swirl, the vertical plane has been rotated by 10 degrees in the swirl direction in order to keep the structure of fuel vapour and combustion in view. The solid line on the horizontal plane illustrates the position of vertical plane, while the position of horizontal plane is indicated on the vertical plane. The upper scale limit of the fuel mass fraction has been set as 0.121, i.e. the fuel equivalence ratio of about 2.

Several features were noted from Fig 5:

1. The strong swirling flow produces effective mixing all the way through. In the early stages of the spray development, fuel is mixed with the surrounding air almost immediately after been evaporated. As the result, the initial vapour clouds are of almost stoichometric fuel/air ratio in the centre.

2. The initial flame occurs from the centre of the vapour clouds. The early stages of combustion have certain characteristics of premixed burning.

3. The spray and flame reach the edge of piston bowl very quickly. Liquid droplets accumulate near the wall and are expected to form wall film. This stage of combustion is more controlled by mixing and could be even more precisely simulated when a more detailed wall film model is employed.

Figure 6 shows the predicted and measured NOx emissions as a function of injection timing. The engine test results show that the NOx emission decreases with the delay of injection. This trend has been well predicted by the CFD simulation. Regarding the NOx level in individual cases, we can see an over prediction of the NOx emission for early injection cases, good agreement or slight under prediction for baseline case and late injection cases. The general agreement between prediction and engine tests has been encourage.

NOx formation is highly sensitive to temperature and also effected by species concentrations. It is no doubt that a precise estimation of flame temperature and a good description of reaction chemistry are essential. However, uncertainty exists regarding the value of NOx scaling factor used in the simulations. More simulation exercises are considered as necessary in order to find a more suitable value of NOx scaling factor to be used with RTZF combustion model.
Figure 5  Case 3: spray, temperature and fuel mass fraction distributions
CONCLUSIONS

CFD simulations of HSDI diesel engine combustion have been conducted to examine the performance of the Ricardo VECTIS CFD code with the RTZF combustion model. Simulations have been performed over a complete injection timing swing.

Good agreement of simulated and measured in-cylinder pressure has been achieved in the combustion simulations, indicating that the VECTIS CFD code with RTZF combustion model is quite capable to cope with DI diesel engine combustion which is typically under a partially mixed condition. The detailed time history of spray, fuel mass fraction and temperature distributions provided by the CFD simulation are valuable towards gaining a better understanding of the features of combustion for different engine configurations under different operating conditions.

The prediction of NOx emissions has been encouraging. CFD simulation has shown a decreasing NOx emission with the delay of injection. This agrees very well with the engine test cases. In the early injection cases, an over prediction of NOx emission is observed, while for the baseline and late injection cases good agreement or slight under prediction have been achieved between the predictions and measurements. It is considered that further investigation might be necessary to find a more suitable NOx scaling factor for use with RTZF combustion model.

ACKNOWLEDGEMENT

The authors would like to thank the Directors of Ricardo for permission to publish this paper.

REFERENCES