Chapter 7
Summary and Conclusions

This project aimed to study the spray dynamics of Heavy Fuel Oil (HFO) that is used in low to medium speed marine diesel engines. This was part of a larger project which aimed to fundamentally understand the properties of HFO spray combustion, in order to improve the operational reliability of low to medium speed diesel engines and to reduce their emissions.

The current project successfully utilised experimental and numerical methods to understand the dynamics of a relatively long duration sprays, including the effects of high fuel viscosity. The project was broadly made up of three parts, namely:

1. Evaluation of individual diesel spray models used in CFD modelling (see Chapter 3),
2. Experimental studies of a diesel spray in a high pressure spray chamber (HPSC) (see Chapter 4), and
3. CFD simulation of a diesel spray using Large Eddy Simulation as the turbulence model for the background fluid (see Chapter 5).

In the first part of the project, simplified numerical simulation cases were carried out to evaluate the sub-models used for a full diesel spray simulation. The Lagrangian-Eulerian multiphase modelling technique was used throughout this project. This part of the project was done prior to the completion of the HPSC experimental set-up.

These evaluations were made to understand more clearly each model’s properties under typical diesel spray conditions. In each evaluation, the models were isolated so that only the effects from the model of interest were shown in the results without unwanted influence of other models. Experimental data from the literature was used to quantify the models’ ability to replicate the physics of a diesel spray.

The evaluation resulted in the following conclusions:

1. Evaluation of the atomisation models found that all three models (Huh and Gosman, MPI and Blob) had the potential to be used in the full numerical simulation. This was because the breakup models and the collision model could influence the dropsize and thus all three models were not ruled out. As a result, all three atomisation models were

- 277 -
2. Evaluation of the droplet breakup models found that most of the breakup models did not completely model the effect of the fuel viscosity. The KH wave breakup model and the KH-RT breakup model were found to be used frequently in diesel spray research publications and thus were programmed into Star-CD for evaluations. The Reitz-Diwakar breakup model, the KH wave breakup model and the KH-RT breakup model were evaluated against the published data of Park et al. [74]. It was concluded that the Reitz-Diwakar breakup model and the KH-RT breakup model had the potential to be used in the full numerical simulation.

3. Evaluation of the droplet drag models found that the default droplet drag model was oversimplified and not capable of modelling a diesel spray accurately. The droplet drag model had to be modified to improve the droplet’s response to the background fluid’s flow direction change. As a result, the TAB based dynamic droplet drag model was programmed into Star-CD.

4. Evaluation of the droplet collision models found that the default model was highly mesh size dependent and required correction to improve the theoretical accuracy. This resulted in the development of the mesh independent O'Rourke collision model (MIOC) which removed the mesh size dependency of the original model.

5. Evaluation of the Lagrangian phase/Eulerian phase coupling models found that the most suitable coupling method was the vertex coupling method.

These findings narrowed down the model combination possibilities for the final numerical simulation found in phase three of the project (see Chapter 5).

The project’s second phase involved conducting experiments using a custom-built High Pressure Spray Chamber (HPSC) at the National Centre for Maritime Engineering and Hydodynamics (NCMEH) at the Australian Maritime College (AMC), University of Tasmania (UTAS). This chamber was specifically designed to study long duration sprays with high viscosity fuel. A constant volume chamber was chosen instead of a piston crankshaft system (like in a real engine) because the chamber permits clear observation of the spray with a large window area, and viewing from various different directions was easily achieved. The chamber also allowed the spray to be injected into still gas without the influence of external turbulent gas flow prior to the start of fuel injection. Instrumentation was supplied by LaVision and included a dual-frame CCD camera with lenses, a double-pulse 120mJ laser with lightsheet optics, a
high-efficiency diffuser, a long-distance microscope and an acquisition computer. The experiment was broken into three parts, namely: spray penetration and cone angle; lightsheet Particle Image Velocimetry (PIV) and dropsize shadowgraphy using a long-distance microscope. The shadowgraphy experiments also allowed application of Particle Tracking Velocimetry (PTV) and micro PIV (μ-PIV). These three experiments allowed the acquisition of the macrostructure of the spray, the velocity profile in the sparse region of the spray and the dropsize in the sparse region of the spray.

The penetration and cone angle results showed evidence that surface instabilities from the shearing of the jet against the surrounding gas and the in-flow of gas into the low pressure region of the spray jet, strongly influenced the overall spray structure. The experimental results confirmed that the penetration rate was a linear plot on a log-log scale (i.e. a power law). The cone angle appeared to have a transition phase from a narrow angle to a wide angle when the breakup process was complete. Then after a certain width, the spray angle appeared to stop expanding radially. This reduced the average overall cone angle. The experiment confirmed that increasing gas density resulted in a lower penetration rate and larger cone angle. The increase of fuel viscosity had a negligible effect on the spray penetration but increased the cone angle.

The PIV measurements were performed on the spray droplets because it was difficult to externally introduce seeding particles. As a result, it was not possible to adjust the seeding particle population density directly. The measured droplet velocities could be considered equivalent to the gas velocities in the sparse region of the spray, because the numerical results suggested there was a minimal difference between the gas and droplet velocity. It was found that the diesel fuel spray was too dense and this contributed to significant errors in the PIV measurement. On the other hand, the 75% canola oil spray PIV results were comparatively more accurate and were confirmed to be sufficiently accurate using PTV. As a result, the PIV measurement analysis was done on the 75% canola oil spray only.

The spray images and the macro PIV analysis showed the presence of high velocity regions and the presence of droplet clusters. For 75% canola oil, the results showed that the bright regions in the scattered light images, which represented regions of high droplet density, tended to correspond to regions of high velocity. The darker regions in the scattered light images, or sparse region, corresponded to lower droplet density and lower velocities. The 75% canola oil spray data showed qualitatively that the highest vorticity magnitude tended to occur at the periphery of the dense regions.
Summary and Conclusions

There was significant variability in spray structure, and velocity from shot to shot. The variability was probably associated with large-scale turbulent fluctuations.

For diesel fuel, the measured maximum mean velocity from macro PIV did not occur at the measurement point on the spray axis closest to the nozzle. This is thought to be attributable to the higher spray density masking the core. Higher velocities were present and were detected only on occasions when droplet density between the light sheet and the camera was low. In contrast, the maximum mean velocity of the 75% canola oil did occur at the closest point to the nozzle on the spray axis. Larger droplets were present due to higher fuel viscosity. These larger droplets offered clear images for the PIV algorithm, even in areas of high spray density.

The velocity RMS profile revealed that there was higher turbulent mixing at the front tip and the radial periphery of the spray. This suggests that the turbulent mixing was higher in those regions.

The dropsize shadowgraphy results provided valuable dropsize data and confirmed that high fuel viscosity had a significant effect on the dropsize of the spray. The results revealed that different chamber pressures did not change the SMD significantly. In fact, the differences were within the bounds of measurement errors. The dropsizes were all measured at 5 ms after start of injection. They showed that SMD of the diesel fuel spray did not change significantly with changing chamber pressure. For chamber pressures of 20 bar, 30 bar and 40 bar, the SMD for diesel fuel was around 22 to 24 µm. The arithmetic mean diameter ranged from 13 to 14 µm. The dropsize of 75% canola oil was measured at 30 bar only, and gave SMD=34 µm and arithmetic mean of 18 µm. The diesel fuel had a dynamic viscosity of 0.0022 kg/ms and the 75% canola oil had a dynamic viscosity of 0.042 kg/ms.

The dropsize shadowgraphy experiment provided point velocity measurements that could be analysed using Particle Tracking Velocimetry (PTV). The PTV measurements were considered more accurate than the PIV measurements because they did not depend on optimum seeding density. Interframe time was optimised so that the movement of individual particles between frames was clearly identified. The comparison between the PTV and the µ-PIV results shows no clear pattern. In most locations, they compare well, but in some locations there is a significant difference. The probable source of the differences is non-optimal seeding density for the µ-PIV. The seeding density was low. In the comparison between PTV and macro PIV, it was found that in the dense region of the diesel fuel spray the macro PIV values were significantly lower than the PTV values. This confirmed that the macro PIV of the diesel fuel spray was erroneous in the dense region.
For the 75% canola oil spray, the microscopic measurements were done at a higher magnification and thus there were insufficient droplet images captured in each frame for μ-PIV. Thus, only PTV and macro PIV were compared. The 75% canola oil spray results showed minimal differences between measurement methods.

The results provided valuable data that were used for validating the full simulation results.

The third phase of the project involved the full numerical simulation of the spray and validation with the HPSC experimental results. A reduced list of models was evaluated under the full numerical simulation set-up based on the knowledge gained from the first phase of the project. The full simulation results were compared with the experimental data from the second phase. The outcome of the evaluation showed that one combination of models produced the closest results to the experimental data:

1. Blob atomisation model (Reitz-Diwakar atomisation model)
2. KH-RT breakup model
3. TAB based dynamic drag model
4. MIOC collision model
5. LES with k-Φ SGS model
6. Vertex interpolation method

This numerical simulation setup was further tested at different gas pressure and validated against the experimental results. It was found that the numerical simulation had very similar penetration rates compared to the experimental results, but the cone angle of the numerical simulation was narrower compared with the experimental results. The PIV measurements and simulation velocity profiles showed similar velocity patterns and velocity magnitudes in the sparse region of the spray. In the dense region, the velocity patterns remained similar but the magnitude of the predicted spray velocity was higher. This was because the experimental results were erroneous in the dense region, due to the higher particle density leading to multiple scattering. The dropsize validation showed that the numerically predicted dropsize was slightly larger than the experimental results, but consistent under all different conditions.

With a successful simulation set-up, it was possible to conduct a more in-depth study of the spray. There appeared to be two regions in the spray: a disintegration region and a stable region.

In the disintegration region, the Reynolds number and Weber number were high. Most of the momentum was transferred from the droplets to the gas phase. In this region, the majority of the
breakup occurred and the collision rate was highest. Shear layer instability occurred at the nozzle exit and formed a turbulent flow structure. There was a presence of a spray core that remained largely intact even though it was wavy in shape due to turbulent mixing and viscosity of the gas fluid.

In the stable region, the Reynolds number and Weber number were low and the droplets scattered outwards to expand the spray volume. This region showed the spray core was broken up into cluster of droplets. The clusters expanded and diluted and led the penetration of the spray. Droplet clusters were apparent within the spray frontal region. There also appeared to be preferential droplet size distribution within the clusters i.e. the larger droplets tend to be at the cluster front whereas the smaller droplets trailed behind. On the spray periphery the chamber gas was entrained by the spray, feeding its radial expansion.

The full numerical simulation results showed that a high viscosity fuel spray contained a significantly different internal structure compared to a low viscosity fuel spray. This was also partly supported by the experimental results. Fuel fractional volume plots of the simulation results were used to qualitatively describe the clustering phenomenon in three dimensions. They showed that the high viscosity fuel spray droplet dispersion rate was significantly lower, and the formation of droplet clusters occurred much further away from the nozzle when compared to low viscosity fuel sprays. The results also showed that an increase in gas density shortened the length between cluster formations.

The outcome of this project is improved understanding of long duration, high fuel viscosity diesel sprays and their simulation with CFD using Large Eddy simulation as the turbulence model for the background fluid. Spray models originally developed for use with Reynolds Average Navier Stokes turbulence models were evaluated in the LES environment. It is concluded that the use of LES as the turbulence model produces good qualitative internal spray structures that predicts the instantaneous turbulent jet structure and the formation of droplet clustering. The project highlights the limitations in the current state of numerical prediction methods and recommendations are made for future work to improve numerical predictions.

### 7.1 Conclusion Overview

What is new in the current work:

1. The current work evaluates all Lagrangian sub-models in an LES environment before using them for the purpose of studying spray structure. This ensures that simulations
were made in the most realistic manner possible and the setup has the flexibility to transfer to different conditions such as different fuel types, different gas pressure, temperature, density, etc.

2. Implementation of a KH-RT breakup model, dynamic drag model and MIOC model into Star-CD.

3. The formulation and use of MIOC in diesel spray simulation.

4. The use of full scale LES in a long duration diesel spray that utilises evaluated Lagrangian sub-models such as breakup models, collision models and dynamic droplet drag model to replicate the experimental results.

5. Fuel viscosity was taken into consideration and has shown to have significant effects on the dropsize and the internal structure of the spray.

6. PIV and dropsize shadowgraphy were used to study the internal structure of the spray and to make comparisons with numerical simulation.

7. A data analysis methodology was developed using DaVis and Matlab to study the experimental results.

What is confirmed in the current work:

1. The default O'Rourke model is mesh size dependent.

2. It is necessary to use a proper dynamic drag model to ensure that the momentum is transferred accurately between the two phases at high Reynolds number.

3. For high fuel viscosities, changes in the fuel viscosity significantly influence the breakup process and an appropriate breakup model is required for accurate simulation.

4. The fuel viscosity plays a vital role in the diesel spray and ultimately in the combustion process.

5. LES produces instantaneous spray structure that is not replicated with RANS turbulence modelling.

6. The effect of droplet clustering phenomena is shown in both experimental and numerical results. This phenomena shows that the spray droplet distribution is not always homogeneous.

What is questioned from the current work:
Summary and Conclusions

1. There is little understanding of the spray cone angle narrowing effect and no proven mechanism has been shown in the thesis on why this occurred.

2. The spray cone angle narrowing effect was not observed in the simulation result. Further work needs to be done to find out why.

3. The lack of mesh size resolution at the nozzle exit region is a source of error but how significant it is to the accuracy and physics in the numerical simulation needs further investigation.

7.2 Recommendations for future work

The current project created an improved solution for the prediction and understanding of turbulent Diesel spray. There are three possible directions for future work, namely, further improvements on current CFD simulation, obtaining more comprehensive experimental results of the spray dynamics and expansion of the current work to study evaporation.

In the area of improving current CFD simulation there are two potential directions. One area involves improving the accuracy of predictions in the nozzle and in the nozzle exit region. It is possible to overcome the poor resolution in these regions via the use of Eulerian/Eulerian multiphase simulation, including cavitation.

Further work would apply the optimised CFD simulation using LES, to actual engines, where the spray interacts with the air motion in the cylinder and with the combustion chamber geometry. As an intermediate step, the HPSC could be used to study the interaction of sprays with representative geometry and the data used to validate CFD models using LES.

The project could also go in the direction of obtaining more comprehensive experimental results. In the current project, many of the experiment results were taken at a specific time. Time resolved spray imaging and PIV could yield more information on evolution of spray structures. However, the spray duration is very short so a very high speed camera would be needed for meaningful time resolved PIV.

Work could be done to more comprehensively quantify droplet clustering for both the experimental results and the CFD results. Better understanding of the fuel droplet spatial distribution in real sprays would be useful in its own right. It would also be valuable to develop methods to quantitatively assess how well the CFD models predict the spatially non-homogeneous fuel distribution in the spray.

- 284 -
The HPSC could be further developed to allow measurements on evaporating sprays. This would allow validation of CFD simulations which include evaporation.