

Direct Injection of Natural Gas/Liquid Diesel Fuel Sprays

T.R White, B.E. Milton and M. Behnia

School of Mechanical and Manufacturing Engineering
The University of New South Wales, Sydney, NSW, 2052 AUSTRALIA

Abstract

Dual-fuelling provides a suitable method for operating compression-ignition engines on alternative fuels. The CFD package *Fluent* is being used to model the direct-injection of two such fuels simultaneously into an engine. The CFD models have been initially calibrated using high-speed photographic visualisation of the jets. Different orientations and staging of the jets with respect to each other are now being simulated. Salient features of the two fuel jets are being studied to optimise the design of a dual-fuel injector for compression-ignition engines. Analysis of the fuel/air mixture strength during the injection allows the ignition delay to be estimated and thus the best staging of the jets to be determined.

Introduction

The introduction of alternative fuels such as natural gas (NG), liquefied petroleum gas (LPG) and alcohols for internal combustion engines is likely to occur at an increasing rate. The dual-fuel (DF) concept allows these low cetane number fuels to be used in compression-ignition (CI, diesel) type engines. Most CI engine conversions have pre-mixed the alternative fuel with air in the intake manifold whilst retaining diesel injection into the cylinder. The advantage is simple adaptation but the main disadvantage is that good substitution levels are only obtained at mid-load. This is because at low load, conventional diesel injectors still require a substantial fuel delivery to operate efficiently while at high load, the extended ignition delay and mixed fuel in the end zone exacerbate both diesel and SI type knock. Other disadvantages are that displacement of air in the intake can reduce peak power at any given fuel/air ratio and that fuel can directly short-circuit from inlet to exhaust, thereby increasing UHC emissions.

A solution is to inject both the alternative and diesel fuels directly into the cylinder. Here, the fuel in the end zone is limited and the diesel, injected before the alternative, has only a conventional ignition delay. This improves the high end performance. Modern, very high pressure diesel injectors have good turndown characteristics as well as better controllability and hence offer an improvement at the low end. Several systems exist, mainly for large marine engines but also for smaller, truck size engines. For the latter, the key is to produce a combined injector to handle both fuels which has the smallest diameter possible so that installation is readily achieved.

This paper details research where the spray characteristics of such a combined injector have been modelled using CFD. CFD allows the details that are difficult to assess experimentally to be quantified. The mixing advantages of different nozzle configurations, the starting interval between and the duration of the flows and the best driving pressures can be ascertained. The computational results are being verified experimentally using high-speed photographic visualisation of the injection of the two streams and their subsequent interaction. Some results from the photographic rig are included here.

Numerical Scheme

The domain used as the basis for the CFD work is based on UNSW's "Rapid Compression Machine" (RCM). The RCM is a constant-volume combustion apparatus and is, in effect, a single-stroke engine simulator [5]. It consists of a cylinder which is the size of that found in an average six or seven litre six-cylinder diesel engine and a piston which can be driven from BDC to TDC at high speed. When the piston reaches TDC it is captured and held, the dual-fuel injector fires and constant-volume combustion occurs. The combustion chamber's pressure and temperature history immediately following the injection is logged for later analysis of the combustion.

Whilst most of the validation of the CFD work will be carried out using high-speed photography, combustion testing of the DF injection will eventually be carried out in the RCM and so the RCM's cylinder was decided upon as the choice for the computational domain. The RCM's simple combustion chamber has a bore of 108mm and a height of 10 to 15mm, depending on the compression ratio used. The computational domain for the results presented here is for the 15mm case which equates to the RCM's lowest compression ratio of about 14:1.

The diesel injector used in these experiments is a mini-sac design with six equi-spaced holes and the gas injector has also been made with six holes. The RCM's cylinder is itself axis-symmetric. Thus, when the two nozzles are oriented so that the holes are co-planar, it is computationally-efficient to model the combustion chamber as a segment of one-sixth of the cylinder.

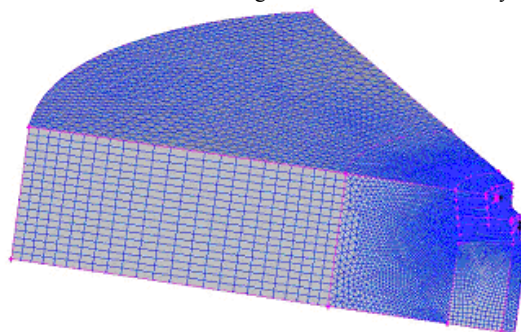


Figure 1. The computational domain used in this study.

Diesel Injection Model

Fluent 6.1.18 is the CFD code being used in this study. *Fluent* allows the modelling of the high-speed injection of liquid into a gaseous atmosphere using a "Plain Orifice Atomiser" (POA) model. This model can be used to simulate a diesel injector which is essentially a long, thin orifice that connects a high-pressure reservoir of diesel fuel within the injector to the compressed air in the combustion chamber. In such an atomiser, the liquid is accelerated through a nozzle, forms a jet and then droplets. Using the POA model enables the injection and subsequent atomisation of the diesel jet to be modelled without

the need to create complicated mesh geometry. The location of the actual orifice and the direction it faces are specified and then parameters particular to the nozzle being modelled are entered. The main parameters used in the case modelled here are as follows:

Nozzle diameter:	0.2mm
Nozzle length:	0.8mm
Chamber temperature:	300K (ambient conditions)
Mass-flow rate:	10.36g/s
Injection duration:	425 μ s

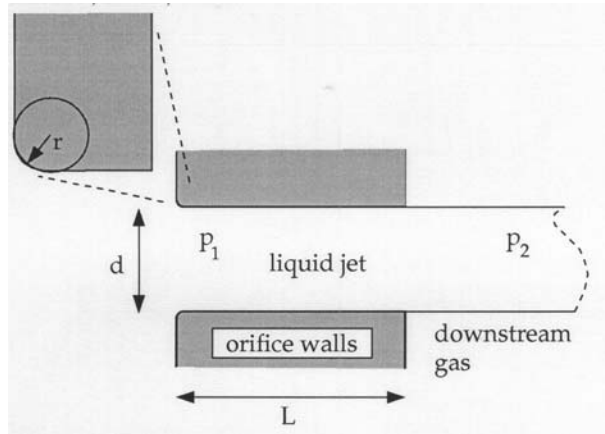


Figure 2. Illustration of the parameters used in *Fluent's* Plain Orifice Atomiser model [1].

Diesel Spray Break-Up and Transport Models

The diesel nozzles being used in this study operate at very high pressures and other researchers support the belief that modern, multi-hole sac-type nozzles usually operate in the cavitating regime [8]. The diesel injector being used in the laboratory work for this study is from a Common-rail, Direct-Injection (CDI) system. It has a multi-hole, mini-sac nozzle and the injection pressure can be set up to 1,800bar. For simulating this nozzle in the CFD, the inlet radius “r” for *Fluent's* POA model was set to a value of 25 μ m which caused the orifice to cavitate.

Fluent offers two spray break-up models: the Taylor Analogy Break-up (TAB) model and the “wave” model. The wave model of Reitz [6] considers the break-up of an injected liquid to be induced by the relative velocity between the liquid and gas phases. This model is popularly used in high-speed fuel injection applications [4, 2] and so is the model chosen for this study. Calibration constants B0 and B1 have been set to 0.61 and 0.20 respectively as suggested by Hong, *et al* [2]. Once the spray has broken-up into droplets, the penetration of the spray depends mainly on the aerodynamic drag of the droplets. *Fluent's* “dynamic drag” model accounts for the effects of droplet distortion by linearly varying the drag between that of a sphere and that of a disc depending on the environmental conditions.

The injection of both the diesel and NG both occur under high pressures and speeds and as a result the jets have high Reynolds' numbers. The “Realisable” k- ϵ model has been chosen for this study since, according to the *Fluent* manual, it more accurately predicts the spreading rate of the round jets being studied here.

Natural Gas Injection

Since the NG does not change its state during or after the injection, its introduction into the computational domain is

through a simple “mass-flow” inlet. In *Fluent*, parameters for such inlets include the actual mass-flow of the fluid as well as the fluid's stagnation temperature. In this study, the gas jet is taken to be choked at a mass-flow of 4.43g/s and thus the stagnation temperature has been set at 343K.

Calibration of the CFD Models

The CFD models are being calibrated using a laboratory rig designed and built by the first author which allows analysis of the dual-fuel jets using a high-speed imaging system. This “Spray Visualisation Rig” (SVR) consists of supply systems for both the diesel and the NG, a prototype DF injector and the imaging equipment.

Diesel is supplied to the DF injector by a pump and accumulator set-up which is based on components taken from a modern CDI system on a commercial vehicle. An electric motor is used to drive the pump and in the absence of the donor vehicle's engine management system, a needle-valve and pulse generator are used to control the rail pressure and injection timing respectively. NG is supplied from a cylinder via a regulator.

The diesel part of the DF injector consists of a CDI injector taken from the same vehicle as the diesel supply system. The gas nozzles used in this study fit co-axially with the diesel nozzle as a DF injector would be fitted in an engine (figure 3). Control of the gas flow is achieved using another modified CDI injector placed in-line with the gas supply to act as high-speed valve. Extensive calculations were performed during design of the gas valve and nozzles to ensure that the flow would be choked and thus the flow rate dependent only on time.

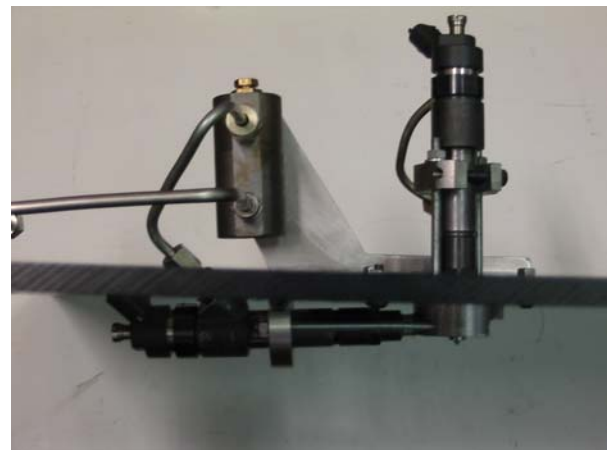


Figure 3. The DF injector assembly. The CDI diesel injector is vertical and fits into the gas nozzle. A second CDI injector (horizontal) controls gas flow into the gas nozzle.

The DF injector has been fitted to a Perspex box to enable photographs of the spray to be taken. Since the NG jet is invisible to the naked eye, special photographic techniques must be employed to capture its image. At first, shadowgraph images were taken using the set-up shown in figure 4. More recently, this shadowgraph equipment has been replaced by schlieren equipment which provides better pictures of the NG jet.

The main parameters used to compare results from the SVR to the CFD are the penetration and shape of the jets at a given time after their respective Start Of Injection (SOI). Whilst penetration of the diesel jet depends upon both its injection pressure and its subsequent atomisation, a certain jet penetration for a given injection pressure has been taken to validate the amount of atomisation in the CFD.

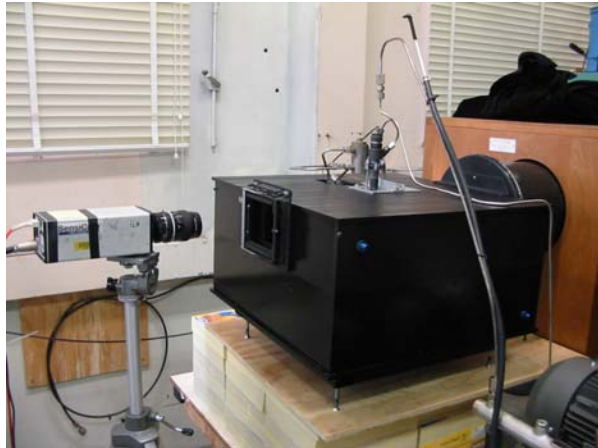


Figure 4. The DF injector fitted to the SVR.

Results

The case being modelled in the CFD work is for an engine of the equivalent size of several RCM-sized cylinders. This engine is assumed to be running with 75% NG substitution at a total fuel/air equivalence-ratio of 0.75 (ie. close to full-load for a diesel engine). For these conditions, the DF injection parameters are as follows:

Diesel Start/End of Injection (S/EOI):	0ms, 370 μ s
NG SOI/EOI:	200 μ s, 4,100 μ s
Diesel nozzle diameter and mass flow:	0.2mm, 10.36g/s
NG nozzle diameter and mass flow:	0.4mm, 4.43g/s

The diesel and gas nozzles are parallel to each other at 12.5° below horizontal. The diesel injection occurs at 1,600bar which is that found in current-generation common-rail systems. The gas flow simulates that which would occur with an injection pressure of 160bar which is the value suggested by a previous researcher at UNSW [3]. Figure 5 shows the distribution of particles 100 μ s after SOI for the diesel. The view chosen is a two-dimensional slice taken vertically through the centre of the computational domain and both the gas and diesel nozzles. In this plot, the area-density of the particles is representative of the concentration of the liquid diesel in that plane in the cylinder. Figure 6 shows a schlieren image from the laboratory rig of actual diesel injection at the same time (100 μ s after SOI). In comparing the CFD results to the photos, a good correlation between the penetration of the two jets may be seen. Also noticeable is the initial “blooming” of the jet near the nozzle.

One of the goals of this study is to find the optimum “staging” for the two fuel jets so that the diesel pilot can be used to virtually eliminate the ignition delay of the NG. Thus the diesel should be igniting at the time of SOI for the NG. Since *Fluent* can simulate the atomisation and subsequent evaporation of the diesel, a plot of the mass-fraction of the diesel vapour relative to the air already in the cylinder can be obtained and is shown in figure 7. This feature of modelling evaporation is useful for helping to predict the ignition delay of the diesel and thus make a good estimate of suitable injection staging. Several researchers in the past who have used CFD to model diesel injection have used non-evaporating models [2, 9]. The availability of the evaporating model in *Fluent* is a distinct improvement over those simulations.

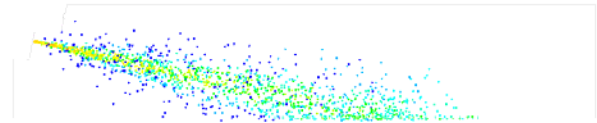


Figure 5. A plot of the diesel particles 100 μ s after SOI, coloured by the speed of the particles where red is the fastest and blue the slowest.



Figure 6. Schlieren image of actual diesel injection at 100 μ s after SOI. The injection pressure is 1,600bar.

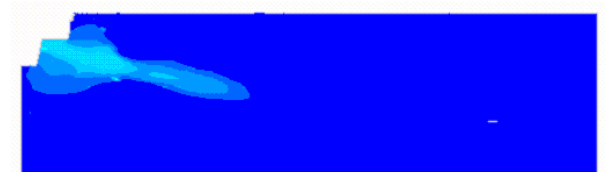


Figure 7. Contour plot of the mass-fraction of diesel vapour in the cylinder at 100 μ s after SOI. The lighter colours represent higher concentrations.

Figure 8 shows a schlieren image of the NG jet by itself at 100 μ s after its own SOI (300 μ s after the diesel’s SOI). Figure 9 shows contours of the mass-fraction of diesel vapour in a diesel-only case at this same time.



Figure 8. Schlieren image of the NG at 100 μ s after SOI. The injection pressure is 160bar.

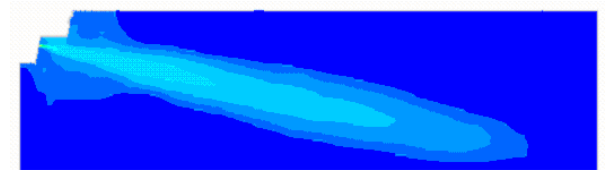


Figure 9. Plot of the contours of diesel mass-fraction at 300 μ s after diesel SOI.

Figures 10, 11 and 12 show CFD results for the DF case captured 300 μ s after diesel SOI. Figure 10 shows the contours of diesel vapour whilst figure 11 shows the NG. Figure 12 shows an overlay of the contours of mass-fraction for both diesel and NG. EOI of the diesel occurred shortly after this series of results whilst the NG injection continued for several milliseconds afterwards.

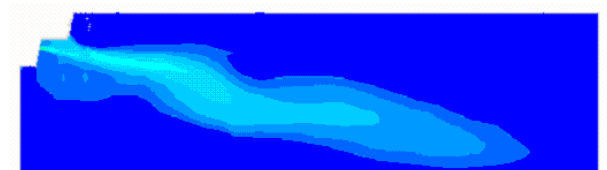


Figure 10. Plot of the contours of diesel mass-fraction at 300 μ s after diesel SOI and 100 μ s after NG SOI. The distorting effect of the NG injection can be seen.

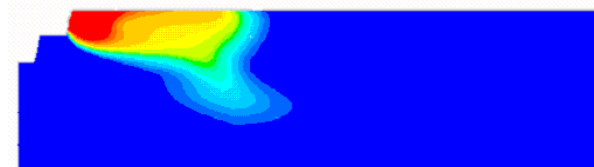


Figure 11. Plot of the contours of NG mass-fraction at 300 μ s after diesel SOI and 100 μ s after NG SOI. Red is a high concentration, blue is low.

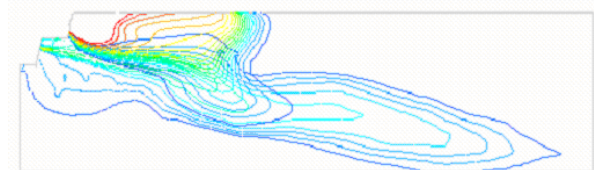


Figure 12. Overlaid contours of both NG and diesel mass-fraction at 300 μ s after diesel SOI and 100 μ s after NG SOI. Significant mixing of the two fuels is clearly evident.



Figure 13. Schlieren image of a laboratory test 300 μ s after diesel SOI and 100 μ s after NG SOI. Diesel and NG injection pressures are 1,600bar and 160bar respectively.

In a real engine, the duration of the gas injection used in the case studied here (4.1ms) may be a little long (one engine stroke at 2,000rpm is 15ms in duration). In a production injector there would be some benefit in making the gas orifices say, 0.6mm in diameter rather than the 0.4mm modelled here. A 0.4mm nozzle was chosen here to enable controlled switching of the gas in the laboratory environment. By making the nozzle only 0.4mm in diameter, the flow time required to deliver the correct mass of gas through the injector was high enough to be achieved accurately. Having said that, there may actually be a benefit in engines for some applications in having an injection as “slow” as that modelled here. This slow injection rate and subsequent burning would help with control of NO_x emissions which often occur as a result of the rapid energy release with modern, high-pressure diesel injectors.

Future Work

Previous studies [7] have shown that the auto-ignition of diesel jets occurs in a diesel vapour concentration band between the equivalence ratios of 1 and 1.5. Subsequent flame development along mixture contours close to stoichiometric then occurs rapidly from there. Thus ignition delay is a function of how quickly the atomised fuel evaporates and forms mixture concentrations conducive to auto-ignition. The evaporation rate depends heavily on conditions into which the fuel is sprayed, especially the temperature and pressure within the cylinder at SOI. The cetane number of the diesel fuel also plays a significant role in cold-start conditions and during warm-up. The staging of the NG injection should therefore be varied depending upon the operating conditions of the engine.

The CFD models developed enable fast and inexpensive testing of operation under these varying conditions. Mesh and time-step independence of the numerical models has not been confirmed at this stage. Such studies are presently being conducted so that

future cases with varying injector orientation and staging can be studied in a way which optimises the computational resources available.

Conclusions

The CFD models created and now running enable identification inside the cylinder of diesel, NG and air mixtures within the flammability range and the number of such potential ignition sites early in the process. The orientation and staging of the gas injection relative to the diesel jet is being varied and the results assessed to find the case whereby ignition delay of the DF injection can be minimised. From there, the final combustion rate is determined for a variety of engine operating conditions.

This data is to be used to find the optimum parameters for the design and development of a production DF injector. Numerical modelling allows quantitative analysis which is not possible using the high-speed photography alone. The laboratory images are, however, being used to validate the numerical work on a qualitative basis using parameters which can be directly compared, ie. spray penetration and shape. Combustion testing in the laboratory will ultimately be carried out using the Rapid Compression Machine.

References

- [1] Fluent 6.1.18 User's Guide, © Fluent Inc., 2003.
- [2] Hong, K.B., Department of Thermal Engineering, ChoongJu University, Korea, Ryou, H.S., Ko, G.H. & Lee, H., Department of Mechanical Engineering, ChungAng University, Korea, The Influence of Breakup Time Constant and Cavitation Modeling on Numerical Simulation of High-Pressure Diesel Sprays, ILASS Asia '00, 5th Annual Conference on Liquid Atomization and Spray Systems, Tsukuba, Japan, 14-15 December 2000.
- [3] Kloeckner, S., Analysis of a Preliminary Design of a Hydraulically-Actuated Natural Gas Injector, Masters Thesis, The University of New South Wales, June 2002.
- [4] Kong, S.C., Senecal, P.K. & Reitz, R.D., Engine Research Center, University of Wisconsin-Madison, Developments in Spray Modelling in Diesel and Direct-Injection Gasoline Engines, Oil & Gas Science and Technology, Volume 54, 1999.
- [5] Mbarawa, M., Problems Related to the Combustion of the Gaseous Fuel (NG) in Diesel Engines; PhD Thesis, The University of New South Wales, 1998.
- [6] Reitz, R.D. & Bracco, F.V., Mechanism of Atomization of a Liquid Jet, Physics of Fluids, 1982, Volume 25, p.1730-1742.
- [7] Rife, J. & Heywood, J.B., Photographic and Performance Studies of Diesel Combustion with a Rapid Compression Machine, SAE Paper 740948.
- [8] Soteriou, C., Andrews, R. & Smith, M., Lucas Diesel Systems, Further Studies of Cavitation and Atomisation in Diesel Injection, SAE Paper 1999-01-1486.
- [9] Tanner, F.X., Swiss Federal Institute of Technology and Paul Scherrer Institute, Liquid Jet Atomization and Droplet Breakup Modeling of Non-Evaporating Diesel Fuel Sprays, SAE Paper 970050.