〈논 문〉 SAE NO. 96370027

새로운 형상의 디젤엔진 연소실 설계를 위한 주위조건의 분석

Analysis Surrounding Condition for the Design of a Novel Direct-injection Diesel Engine Combustion System

박 권 하*, T. F. Yeung**
K. Park

요 약 문

디젤 엔진의 분사연료를 연소실 내부에 마련된 작은 돌출부에 충돌시켜 액적을 작게 부수고 연료가 연소실 내부에 고루 분포할 수 있도록 하여 여러 가지 엔진성능향상을 도모한 새로운 디젤 연소실 시스템이 최근 제시되고 있다.

이들 시스템은 피스톤 내부 혹은 엔진헤드 부위에 분사연료 충돌부를 두고 있는데, 여기에서는 이 새로운 시스템 개발에 있어 고려되어야 할 몇 가지 중요 요인들에 중점을 두어 분석하였다.

결과로서 분사압력, 분사노즐크기, 주위공기 온도와 압력의 변화가 분무 평균입경과 분무연료의 분포에 미치는 영향을 제시하였다.

주요기술용어 : Spray(분무), Diesel engine(디젤 엔진), Injection(분사), Impaction(충돌)

1. Introduction

Over the past years, many research works have been carried out to investigate the factors which govern the performance of diesel engines. The area of study was focused on the combustion chamber. The reason for concentrating on this area is due to the fact that higher efficiency can be obtained by improving the atomization, vaporization, and distribution of the fuel and the extent of mixing with air. As a part of the effort, new geometries using spray wall impaction were proposed by Kroeger[1], Naver et al[2], Kato and Onishi[3] and Ogura et al[4]. Fig.1 shows the geometry by Kato and Onishi[3].

Recently, as shown in Fig.2 a novel design of direct injection diesel engine combustion system was proposed by Park et al[5] and is being developed by them. This involves squirting the fuel onto a flat projecting surface(a

* 한국기계연구원
** UMIST

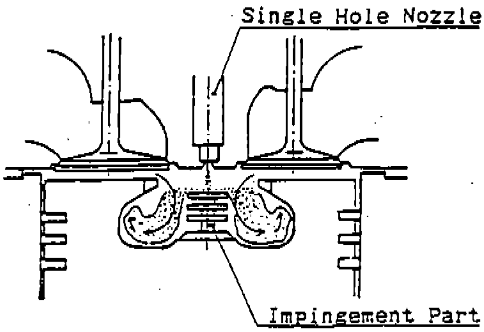

Fig.1 Geometry of OSKA-D system

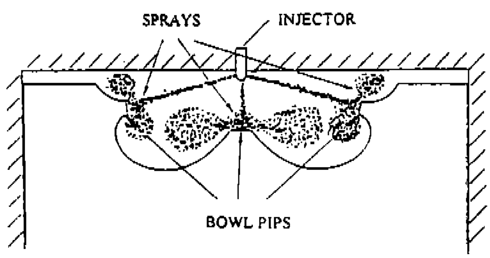

Fig.2 Bowl geometry proposed by Park et al

pip) in order to break the fuel, and to assist atomization, evaporation and combustion. Preliminary calculations indicate that the system has potential to adequately distribute the fuel spray over the piston bowl. The spray is distributed away from the bowl wall and then a good vaporization and mixing with air would be resulted. If that mechanism is possible without resorting to swirling the flow, substantial combustion efficiency gains would be made over conventional high-swirled systems,

This paper investigates the effects of variation in gas pressure and temperature, injection pressure and injector nozzle size by computational approach to help design the combustion chamber shape.

2. Mathematical model

The complete mathematical model employed in computation has been set out in a number of publications. In particular, Watkins[6] and Khaleghi[7] give, in considerable detail, conservation equations for the two phases, the interaction between the phases, and drop breakup. Details of the methods used to solve these are also contained therin. Here therefore only a brief outline is given.

The gas phase is decribed by Euler conservation equations for mass, momentum, energy and fuel vapour mass fraction. The turbulent nature of the flow is calculated by solving additional transport equations for the turbulent kinetic energy and its dissipation rate(the $\kappa-\varepsilon$). These can be witten in the general form;

$$\frac{\partial}{\partial t}(\theta\rho\Phi)+\nabla\cdot(\theta\rho\vec{V}\Phi)=\nabla\cdot(\theta\Gamma_{\Phi}\nabla\Phi)+\theta S_{\Phi}+S_{\Phi}^{d} \tag{1}$$

where ρ is the gas density, $\vec{V}$ is the velocity vector, S_{Φ} is the source term for each Φ which is a conserved property such as u^i, e, f, κ, ε and 1 for continuity equation. The void fraction θ is defined as the volume fraction of gas in each cell, that is

$$\theta=1-\frac{\triangle V_{liquid}}{\triangle V_{cell}} \tag{2}$$

where $\triangle V_{cell}$ is the cell volume and $\triangle V_{liquid}$ is the volume of liquid in the cell.

The liquid phase of fuel is assumed to be fully atomized into spherical drops. The motions of the drops are described by Lagrangian equations for position and momentum. And the states of the drops are also described by Lagrangian equations for droplet size and energy. These equations are given as;

$$\frac{d\bar{r}_d}{dt}=\overline{V}_d \quad (3)$$

$$\frac{d\overline{V}_d}{dt}=k_d(\overline{V}+\overline{V}'-\overline{V}_d) \quad (4)$$

where the fluctuating velocity $\overline{V}'$ is given using the stochastic method of Gosman and Ioannides[8], and is randomly selected with the standard deviation of the Gaussian distribution σ,

$$\sigma=\sqrt{\frac{2k}{2}} \quad (5)$$

where k is the turbulence kinetc energy.

A stochastic approach is adopted for the liquid phase in which only a number of representative drops are calculated for. Each of these drops is held to represent thousands of other drops having the same characteristics of position, velocity, size etc. Full two-way interactions between the phases are accounted for in terms of drag force, evaporation, etc.

The equations for gas phase are solved by finite-volume method. In the discretisation process, the Euler method is empolyed for temporal advancement, resulting in fully implicit equation sets. The hybrid of central and upwind differencing due to Spalding[9] is used for spatial discretisation of convection, while central difference scheme is used for diffusion.

The Lagrangian equations are discretised in time, using Euler method, to give implicit equations to solve for the drop position, velocity, mass and temperature.

The velocity-pressure coupling in the gas phase momentum equations is handled through the non-iterative solution scheme PISO[10]. The introduction of a second phase into the solution is described by Watkins[2].

3. Chamber design and test cases

Throughout the test a constant volume bomb is considered with diameter 100㎜ and depth 20㎜. Fig.3 shows the simulated combustion chamber. Due to symmetry, only half sectional view is required. The piston bowl with a flat-topped pillar in the center of the bomb is simulated, The fuel spray is injected down the center of the bomb from the injector and impacts on the flat top of the pip. The distance from the nozzle to the pip land and the pip radius are set to be 10㎜ and 1.7㎜ respectively, which were given as the conclusion of[5].

The flow is assumed to be axisymmetric and the computational grid adopted is shown in Fig.4, the 23×40 line grid only is used throughout the calculations in order to maintain consistency.

As shown in Table 1, the parameters investigated cover a wide range of conditions of injection radii(0.05-0.25㎜), injection pressures

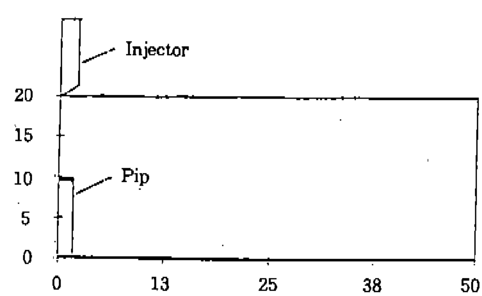

Fig.3 Simulated combustion chember

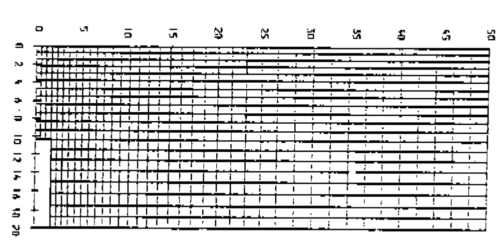

Fig.4 Computational grid

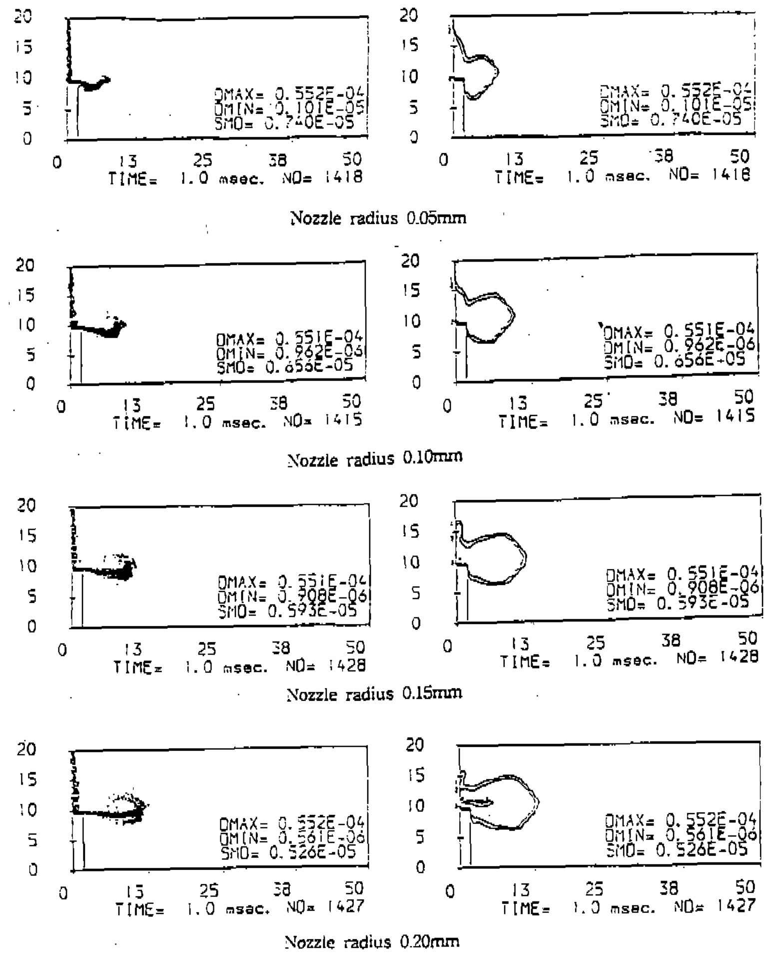

Fig.5 Fuel vapour and spray distribution for nozzle size variation

Table 1 The parameters investigated

Nozzle radius, RINJ(㎜)	0.05, 0.075, 0.1, 0.125, 0.15, 0.175, 0.2, 0.25
Injection pressure, PINJ(bar)	200, 300, 400, 500, 600, 700, 800, 900
Trap pressure, PTRAP(bar)	5, 15, 30, 40, 50, 60, 70, 80, 90, 100
Trap temperature, TTRAP(K)	400, 473, 500, 600, 700, 773, 800, 1000, 1200, 1500

(20-90MPa), gas pressures(0.5-10MPa) and gas temperatures(400-1500K).

4. Results and discussion

In the simulation, the spray is assumed fully atomized into a range of drop sizes between 5 and 55μm at the injector. The Sauter mean diameter(SMD) is specified as 18μm. And the results are discussed with the spray distribution, SMD and volumes occupied by the spray.

Fig.5 shows fuel vapour and fuel spray distributions at 1.0ms after injection for 4-cases out of 8 test cases varing injector nozzle sizes. It is observed that the spray and the fuel vapour spread out quickly with increasing the nozzle radius and make a cloud around the pip. In case of trap pressure a lot of big drops make a lump near the pip side with increasing, which shown in Fig.6. for the other tests the shapes of droplet distributions or vapour contours were given in Yeung[11]. With increasing injection pressure or trap temperature, more atomized droplets after impaction on the land of the pip go into the free space.

The volumes occupied by the spray are shown in Figs.7-10. The volume increases more rapidly than the increasing rate of the nozzle radius, which means the spray injected from very small hole nozzle may loose lots of its energy before it reaches on the land of the pip, so the spray does not spread enough. In case of injection pressure in increases linearly with the pressure increasing, and the volume increases again with trap temperature increase but its rate reduces and the volume is not changed any more in high temperature region above than 1000K. It seems that a number of fuel drops may be evaporated and the

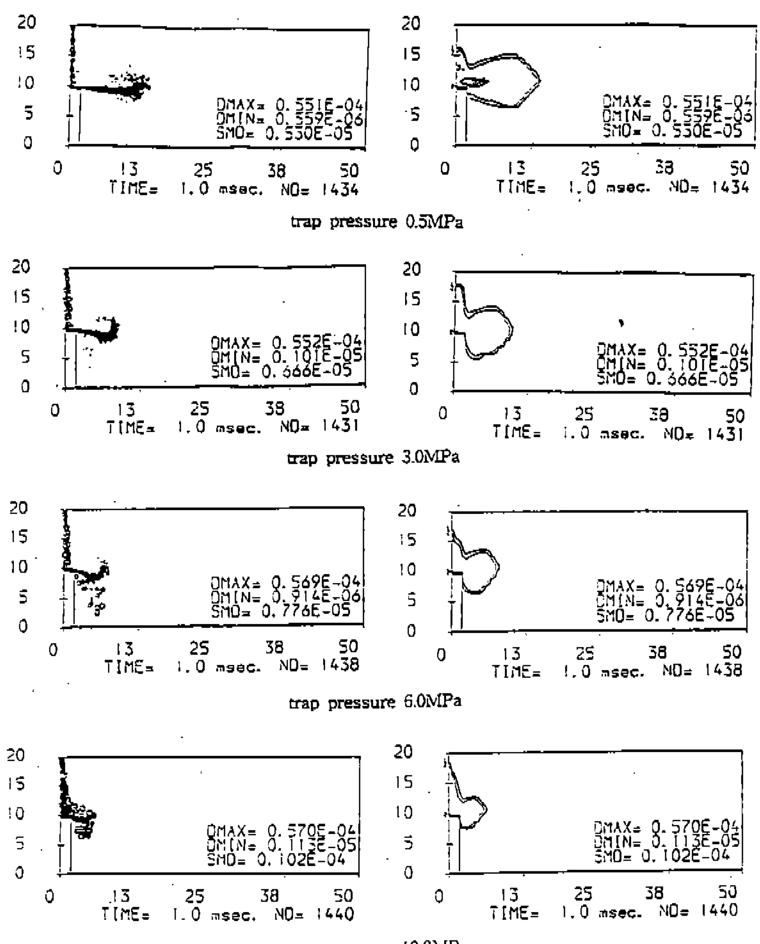

Fig.6 Fuel vapour and spray distribution for trap pressure variation

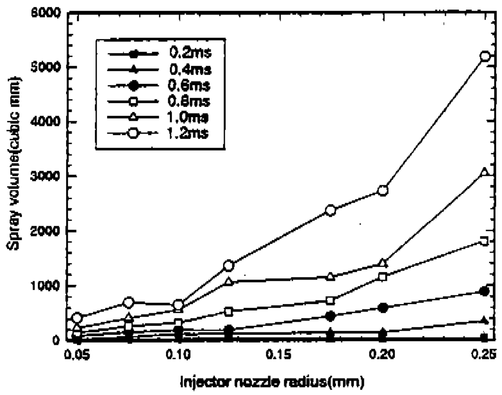

Fig.7 Spray volume against injector nozzle size

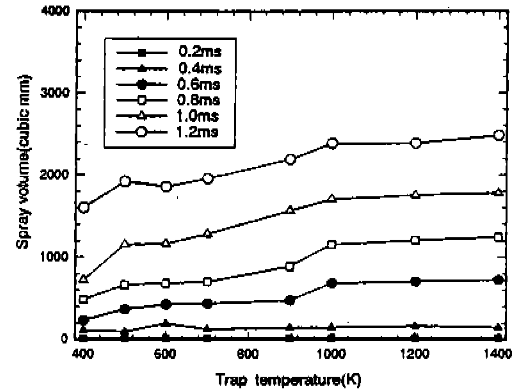

Fig.9 Spray volume against trap temperature

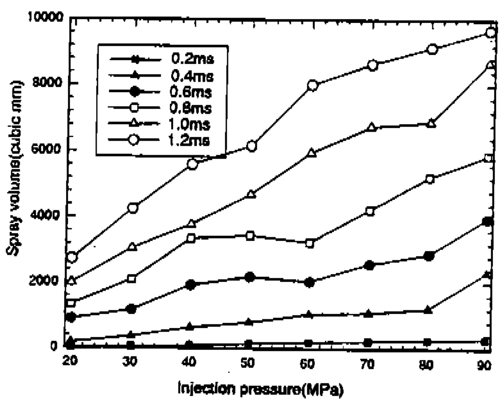

Fig.8 Spray volume against injection pressure

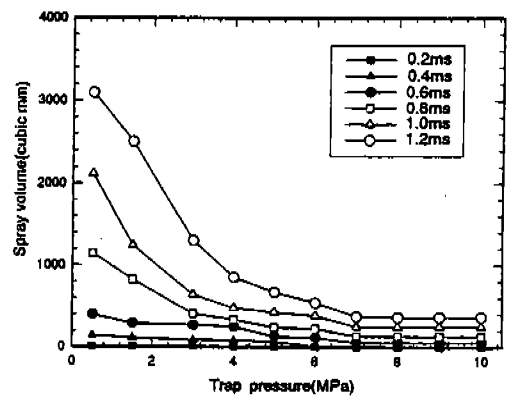

Fig.10 Spray volume against trap pressure

force to push out into the combustion volume bomb may reduce. With the increasing trap pressure the volume occupied by spray decreases but its decreasing rate rapidly reduces so the volume does not decrease in over 70 bar.

Figs.11-14 show SMD against various parameters. SMD decreases with the increase of the nozzle radius, the injection pressure or the trap temperature. But with increasing trap pressure it increases. In case of free spray without any wall impingment, increasing the nozzle hole size makes SMD increasing generally. However in this study the spray injected from the nozzle impinges on the small land and disperses into free space soon after. The spray is atomized by impaction on the wall and leasves the wall without enough time to coagulate each other. Therefore SMD decreases with the increase of the nozzle hole size.

Table 2 summarize the effects of increasing the parameters. From the table, it is observed that, increasing the injector nozzle size, injection pressure and trap temperature all cause a decrease in SMD but only trap pressure gives a negative result. In particular, increasing injection pressure gives a rapid and large effect together with a smallest value of SMD.

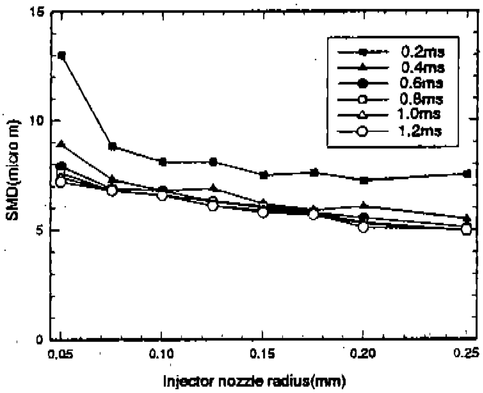

Fig.11 SMD against injector nozzle size

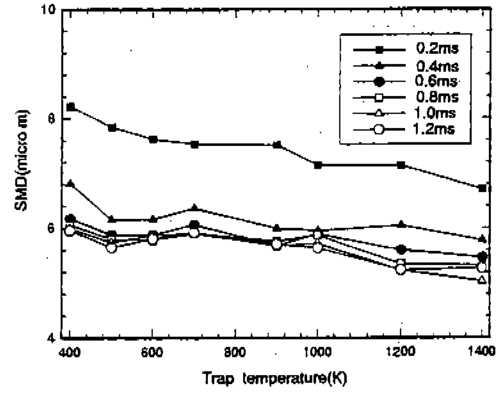

Fig.12 SMD against injection pressure

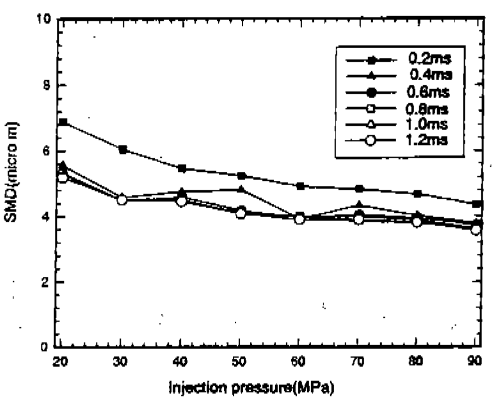

Fig.13 SMD against trap temperature

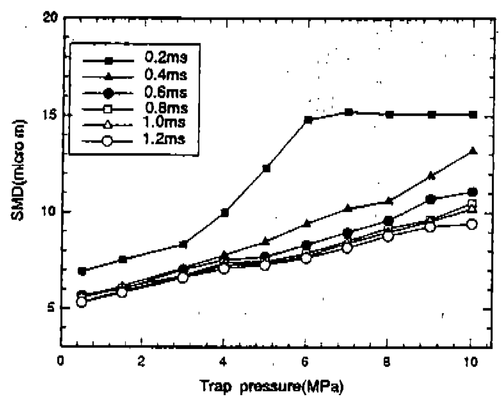

Fig.14 SMD against trap pressure

Table 2 The effects of varying different parameters

Parameters	SMD	Spray volume	Vapour mixture fraction	Smallest value of SMD(μv)
Injector nozzle size	Decrease slightly more	Increase slightly more	Increase slightly	0.495
Injection pressure	Decrease rapidly	Increase rapidly	Increase rapidly	0.363
Trap temperature	Decrease slightly	Increase slightly	Increase significantly	0.530
Trap pressure	Increase	Decrease	Decrease significantly	Greater then 0.530

For the effects on spray volume, the former three parameters can cause an increase in spray volume as it is increased, again only increasing trap pressure gives a negative result. Consider the effects on fuel vapour mixture fraction, the fraction increases significantly with increasing trap temperature, increasing the nozzle size and injection pressure also produce an increase in this value. However increasing the trap pressure significantly decreases the value.

5. Conclusions

Following conclusions for the effects of varying different parameters of the model are drawn.

1) A larger nozzle produce a larger spray volume which covered by a reasonable area of high fuel vapour mixture fraction, increasing this parameter cause a decrease in SMD.
2) The injection pressure(nozzle opening pressure) has a significant effect on both the spray volume and the SMD. The higher the injection pressure, the larger the spray volume and the smaller SMD. Very high degree of fuel vapour mixture fraction can be achieved with high injection pressure.
3) The spray volume increase slightly with the increasing ambient gas temperature up to approximately 1000K, SMD slowly with increasing trap temperature. The higher the gas temperature the larger the area that processing high value of fuel vapour mixture fraction.
4) Increasing the ambient gas pressure leads to a decrease in spray volume. SMD obtained is generally large and increases with increasing gas pressure, only a small area covered by combustible fuel vapour mixture fraction can be obtained and the area have the tendency to decrease heavily as trap pressure increases.

참 고 문 헌

1. C. A. Kroeger "A neat methanol direct injection combustion system for heavy duty applications", SAE Paper 861169, 1986.
2. J. D. Naber, B. Enright and D. Farrel "Fuel impingement in a direct injection diesel engine", SAP Paper 881316, 1988.
3. S. Kato and S. Onishi "New type of diesel engine by impingement of fuel jet(OSJA-D)", SAE Paper 901618, 1990.
4. M. Ogura and B. Lin "A new multi-impingement-wall head diffusion combustion system(NICS-MH) of a d. i. diesel engine", SAE Paper 940196, 1994.
5. K. Park, D. M. Wang and A. P. Watkins "A contribution to the design of a novel direct-injection diesel engine combustion system-analysis of pip size", Appl. Math. Modelling, Vol. 17, pp. 114~124, 1993.
6. A. P. Watkins "Three-dimensional modelling of gas flow and sprays in diesel engines", In Makatos, N. C., editor, Computer Simulation of Fluid Flow, Heat and Mass Transfer and Combustion in Reciprocating Engines, pp. 193~237, Hemisphere, 1989.
7. H. Khaleghi "Three-dimensional modelling and comparison with experiment of sprays and gas flows in test rigs and diesel engines", PhD Thesis, University of Manchester, Faculty of Technology, 1990.
8. A. D. Gosman and E. Ioannides "Aspects of computer simulation of liquid fuelled combustors" AIAA Paper No. 81-0323,

1981.

9. D. B. Spalding "A novel finite-difference formulation for differential equations involving both first and second derivatives", Int. J. Numerical Method in Engineering, Vol. 4-151, 1972.
10. R. I. Issa "Solution of the implicitly discretised fluid flow equation by operator splitting", J. Comput. Phys., Vol. 62-1, pp. 40~65, 1986.
11. Y. F. Yeung "Towards the design of a novel direct-injection diesel engine injection and combustion system", Final year project report, University of Manchester Institute of Science and Technology, 1994.