RESEARCH ARTICLE OPEN ACCESS

Soot Formation in Diesel Engines By Using Cfd

R. Siva Kumar[1], Dr. K. Tirupathi Reddy[2], P. Madhu Raghava[3], P.Nagaraju[4]
*(Mechanical Engineering, Santhiram Engineering College , Nandyal-518501, India.)
**(School Of Mechanical Engineering, Rgmcet , Nandyal-518501, India.)
*** (Mechanical Engineering, Santhiram Engineering College , Nandyal-518501, India.)
**** (Mechanical Engineering, Santhiram Engineering College , Nandyal-518501, India.)

ABSTRACT
In order to meet the stringent emission standards significant efforts have been imparted to the research and development of cleaner IC engines. Diesel combustion and the formation of pollutants are directly influenced by spatial and temporal distribution of the fuel injected. The development and validation of computational fluid dynamics (CFD) models for diesel engine combustion and emissions is described. The complexity of diesel combustion requires simulations with many complex interacting sub models in order to have a success in improving the performance and to reduce the emissions. In the present work an attempt has been made to develop a multidimensional axe-symmetric model for CI engine combustion and emissions. Later simulations have been carried out. Commercial validation tool FLUENT was used for simulation. The tool solves basic governing equations of fluid flow that is continuity, momentum, species transport and energy equation. Using finite volume method turbulence was modeled by using RNG K-ε model. Injection was modeled using La Grangian approach and reaction was modeled using non premixed combustion which considers the effects of turbulence and detailed chemical mechanism into account to model the reaction rates. The specific heats were approximated using piecewise polynomials. Subsequently the simulated results have been validated with the existing experimental values.
Keywords: IC engines, CFD, FLUENT, RNG K-ε model, La Grangian approach.

I. INTRODUCTION

The diesel engine, because of its highest thermal efficiency among currently available engines, has been a main power source for over a hundred years. It's advantage in thermal efficiency is due to its combustion characteristics. Diesel fuel is injected with a high pressure environment (compression ratio as high as 24:1). The higher the compression ratio, the more efficient the cycle [Hey Wood, 1998].

However, for some of the reasons that the diesel engine is highly efficient, its power density and exhaust emissions have traditionally been less desirable than spark ignition (SI) combustion engines. In a diesel engine, the combustion rate is controlled by the fuel injection rate and mixing and diffusion rates, which are usually slower than the premixed combustion rate in typical gasoline engines. Diesel engines usually emit more particulate and NOx than their gasoline court parts the high temperature and pressure environment in the diesel engine cylinder because of its high compression ratio and high combustion temperatures makes it impossible to completely prevent NOx form forming. The soot formed in the fuel – rich regions, although partially oxidized in the expansion stroke, also remains in considerable amounts at exhaust valve opening (EVO). The problem of diesel engine emissions is exacerbated because of the trade - off feature between NOx and soot emissions. It is usually impossible to reduce both kinds of emissions simultaneously, since factors that tend to decrease one usually increase the other. For example, retarding the fuel injection timing is effective to reduce NOx formation by reducing the peak cylinder temperature and pressure. However, this method results in an increase of soot production because more soot formed due to the lower in - cylinder gas temperature has shorter time to be oxidized [Lee, 2002]. Increasing the EGR rate can decrease the NOx emission level, however less oxygen is available to oxidize soot. Eventually, any change in these engine parameters will unavoidably affect other important engine performance measures, like retarding injection timing causes lower thermal efficiency and higher Brake Specific fuel consumption (BSFC) [Hey Wood, 1998].

Increasing environmental concerns and legislated emission standards have led to the necessity of considering both conventional and unconventional means for reducing soot and NOx emissions in diesel engines which is also the motivation of the present study. For example, diesel engine manufacturers are facing the challenges of the extremely low diesel engine – out soot emission mandates to be implemented in the near future. Engine simulation, compared to expensive engine experiments, is an efficient way to investigate various novel ideas to improve current engine performance, and hence becomes an essential part of engine research and development. In addition, simulations can investigate the transient properties of physical process. e.g., diesel sprays combustion and emission formations, whose characteristics are difficult or impossible to obtain with the current measurement technologies.

II. OBJECTIVE AND APPROACH

The objective of this study was to improve diesel engine performance using computational optimization methods. Multiple injection strategies (split injection), with up to 2, 3 and 4 pulses / cycle and 1500 bar to 2500 bar injection pressure, different cone angles, different SOI (start of injection), different DOI (duration of injection) were investigated to minimize engine – out NOx, soot, HC and CO emissions, along with BSFC using the computational models in FLUENT 6.3, and to compare the results with the existing experimental results. The purpose of this optimization was to gain a detailed understanding of the mechanisms through winch fuel injection interacts with other engine parameters and influences diesel combustion and emissions, and hence to attempt to generalize the adoption of multiple injection strategies with regard to improving HSDI diesel engine performance.
Simulations were performed to understand the influence on various parameters on engine performance.

III. METHODOLOGY

The methodology employed in this project was designed to accomplish the objectives described. This is shown pictorially in Fig 3.1 and the purpose behind each of these sections described here in detail.

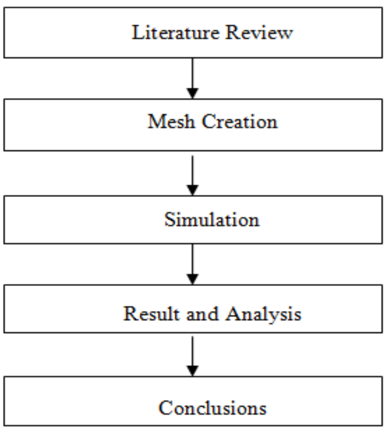


Fig.3.1 Structure of Methodology

IV. SOURCES OF SOOT FORMATION

Fig.4.1 Power Generation

Fig.4.2 Vehicle Exhaust

V. SOOT FORMATION PROCESS

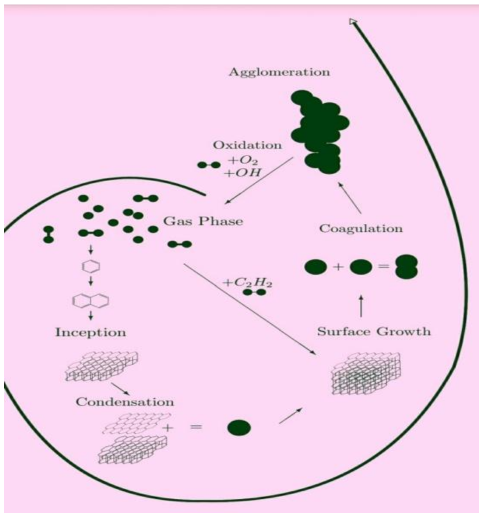


Fig.5.1 Stage-1

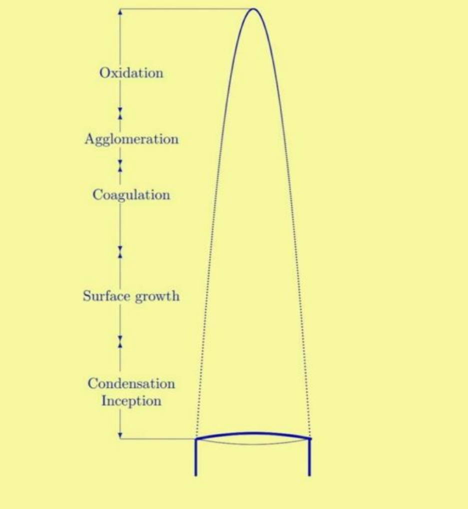


Fig.5.2 Stage-2

VI. COMPUTATIONAL FLUID DYNAMICS

CFD is a sophisticated analysis technique that the analyst to predict transfer of heat, chemical reaction, and fluid flow behavior etc. CFD is based on the fundamental governing equations of fluid dynamics- the continuity, momentum, and energy equation. It is a powerful tool to carry out numerical experiments. This project uses the Computational Fluid Dynamics –FLUENT software package.

The process of utilizing FLUENT can be assumed. Firstly, the geometry and gird is created using GAMBIT. T Grid can be used to generate 2D triangular, 3D tetrahedral or 2D and 3D hybrid volumes mesh from an existing boundary mesh. Another alternative of creating grids for FLUENT is using ANSYS or IDEAS (SDRC); or MSC/ARIES, MSC/PATRAN. Pre BFC and GeoMesh are the names of FLUENT Pre-processors that were used before the introduction of GAMBIT.

Once a grid has been read into FLUENT, all remaining operations are performed with in the solver. These include setting the boundary conditions, defining fluid properties, and material properties, executing the solution, refining the grid, viewing and post- processing the results.

VII. MESH GENERATON

The local flow characteristics of high speed liquids are not necessarily parallel to the flow of direction. Therefore, it is pertinent to use unstructured triangular grids. The commercial mesh generation package GAMBIT is used .GAMBIT uses the Delaunay triangulation automatic mesh generation technique, with an advancing front method. All the meshes are axi-symmetric in two and three –dimensional. The advantages of Delaunay triangulation are that no point is contained in the circum- circle of any triangle and it maximizes the minimum angle for all triangular elements, which is a requirement for good quality finite elements, i.e. the mesh has a low skewness.

The mesh is generated by applying points along the boundaries of geometry with certain stretching function. The geometry made of edges and faces. It was found that geometries, which had interiors that were made of multiple faces, performed better in calculation. This was due to the increase in versatility of the stretching function that minimized a large cell volume jumps within the domain.

The mesh can be modified as a function of the solution. Post solution adaption is undertaken in FLUENT. First, the individual cell s is marked for refinement based on an adaption function, which is created from the solution data. Next, the cell is refined based on these adaption marks. The mesh is adapted by the hanging node technique, whereby the cells marked for refinement are isotropically subdivided. Fig 3.2 shows an example of this technique.

VIII. GOVERNING EQUATION

In the CFD methodology, fluid flows are stimulated by numerical solving partial differential equations that governs the transport of flow quantities also known as flow variables. These variables include mass, momentum, energy, turbulent quantities, and species concentrations. In designing the POME- nozzle, the basic governing equations that will be used are the conservation of mass, momentum and energy equations.

7.1 Continuity Equation

In any fluid flow, the equation for conservation of mass, or continuity equation, can be written as follows:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial xj}(\rho u_j)=\dot{m}$$

Where u_j is the j^{th} Cartesian of the instantaneous velocity, ρ is the density and $\dot{m}$ represents the rate of mass generation in the system. This equation is valid for incompressible as well as compressible flows.

7.2 Momentum Equation

The law of consecration of momentum states that

$$\frac{\partial \rho u_j}{\partial t}+\frac{\partial}{\partial xj}(\rho u_i u_j)=-\frac{\partial \rho}{\partial x_i}+\frac{\partial \tau_{ij}}{\partial x_i}+\rho f_i$$

Where ρ is the static pressure, τ_{ij} is the viscous stress tensor and _ is the body force.

For Newtonian fluids:

$$\tau_{ij}=\mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\mu\left(\frac{\partial u_k}{\partial x_k}\right)\delta_{ij}$$

Where μ is the fluid dynamic viscosity and δ_{ij} is the Kronecker deltas.

Substitute equation 3.4 into 3.3results in the Navier – Strokes Equation

$$\frac{\partial}{\partial t}(\rho u_i)+\frac{\partial}{\partial x_j}(\rho u_i j_i)+\frac{\partial}{\partial x_j}\left\{\mu\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_j}-\frac{2}{3}\mu\frac{\partial \mu_k}{\partial x_k}\delta_{ij}\right\}+\rho f_i$$

7.3 Energy Equation

The energy equation governs the enthalpy balance. For CFD, the static enthalpy is used Conservation of energy is described by

$$(\rho h)+\frac{\partial}{\partial x_j}(\rho u_i h)=-\frac{\partial q_i}{\partial x_j}+\frac{\partial}{\partial \rho}(\rho)+u_j\frac{\partial \rho}{\partial x_j}+\tau_{ij}\frac{\partial u_i}{\partial x_j}-\frac{\partial (J_{ij}h_i)}{\partial x_j}+S_a$$

Where J_{ij} is the total diffusive mass flux for species ai, h_i is the enthalpy for species ai, S_a stands for additional sources due to surface reaction, radiation and liquid spray and q_j is the j component of the heat flux. Fourier_ Law is employed to model the flux

$$q_k=-K\left(\frac{\partial T}{\partial x_j}\right)$$

Where K is the thermal conductivity. This is only true for incompressible at low Mach number. The total enthalpy of the energy equation is fully conservative and is recommended for high –speed compressible flows. The total enthalpy H is defined as

$$H=h+\frac{u_j u_j}{2}$$

The governing equation for H can be obtained by adding the fluid kinematics energy equation to the static enthalpy equation:

$$\frac{\partial}{\partial t}(\rho H)+\frac{\partial}{\partial x_j}(\rho u_i H)=\frac{\partial}{\partial x_j}(K\frac{\partial T}{\partial x_j})+\frac{\partial \rho}{\partial x_j}+\frac{\partial \tau_{ij}u_i}{\partial x_j}-\frac{\partial (J_{ij}h_i)}{\partial x_j}+s_a+\rho f_i u_i$$

VIII. SIMULATIONS FOR MODEL VALIDATION

This chapter summarizes the simulation results performed by other researchers on various diesel engines, from which the date were drawn for validating the CFD models developed and used in this study. Section 4.1 describes the simulation on the High-Speed Direct – Injection (HSDI) diesel engines.

The engine is a fully-instrumented single cylinder version of the caterpillar 3406 heavy duty truck engine. Simulations were carried at high load where EGR has been shown to produce unacceptable increase in particulate emissions. The fuel system was an electronically controlled, common rail injector and supporting hardware. The fuel system was capable to handle four independent injections per cycle. The spray nozzle angles used with included angle 1250 and 1400.

8.1 SIMULATION SET-UP

The test engine is a fully instrumented single cylinder version of a caterpillar 3406 heavy duty truck engine. The engine is capable of producing 54 KW at a rated spread of 2100 RPM. The fuel was an electronically controlled common rail injector and injection pressure unto 1200 MPa was possible so as to ensure fuel system reliability. The injection system is capable of single, double, triple and quadruple injection schemes. Timing and duration could be varied independently for each injection pulse providing a highly flexible injection system. A 0.6 mm diameter orifice was used to produce very fast injection rates 2 crank angles to maximum needle lift at 1600 RPM.

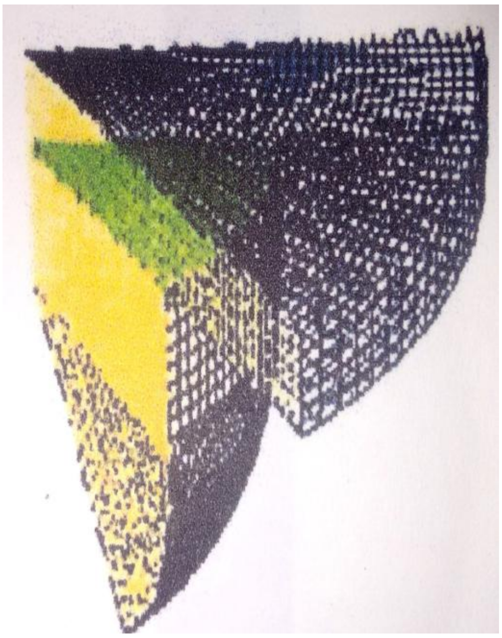


Fig.8.1 Computational Grid for Simulation

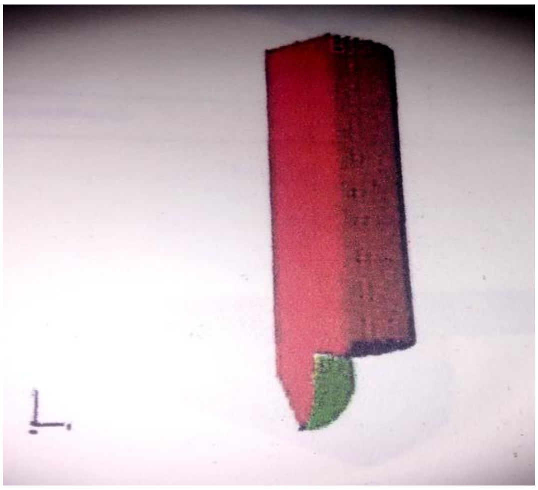


Fig.8.2 Dynamic Mesh of the Grid

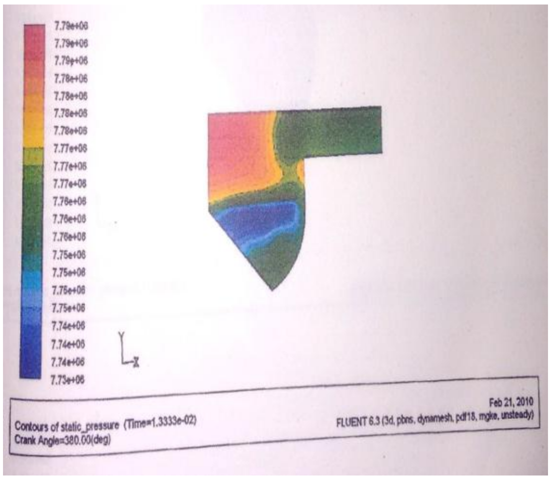


Fig.8.3 Pressure Contours-1

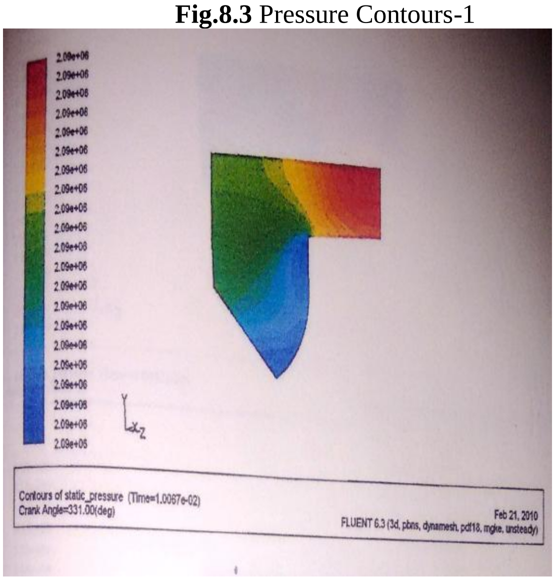


Fig.8.4 Pressure Contours-2

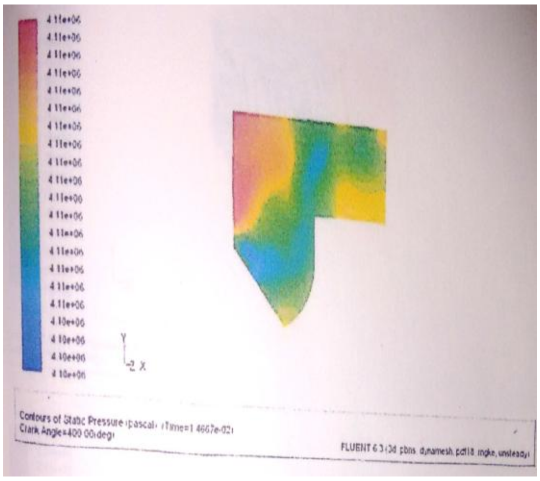


Fig.8.5 Temperature Contours-1

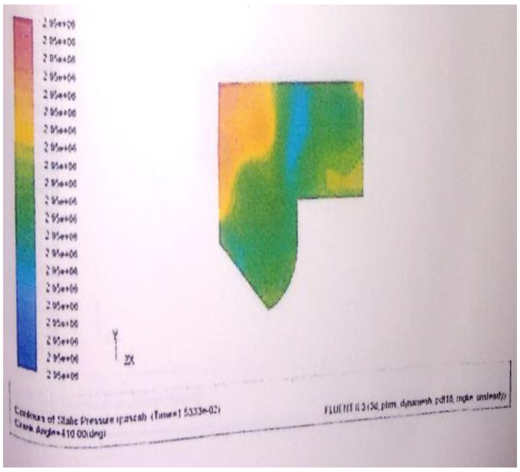
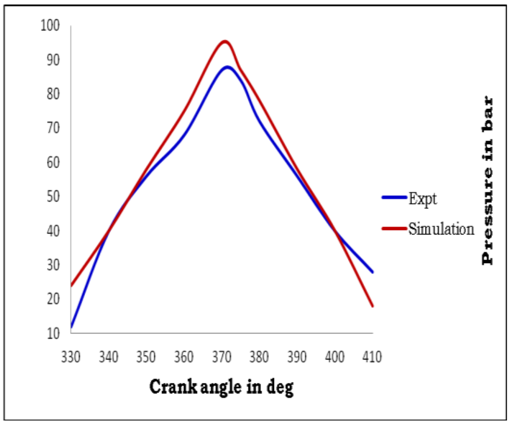


Fig.8.6 Temperature Contours-2

IX. GRAPHS



9.1.Validation For Cycle Pressure

Fig.9.1 Graph: P-θ Curve for single injection

9.2. Validation For No_x

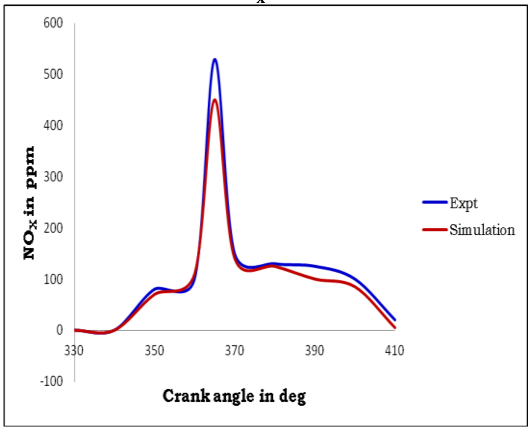


Fig.9.2 Graph: NO_X Curve for Single Injection

9.3. Validation For Soot

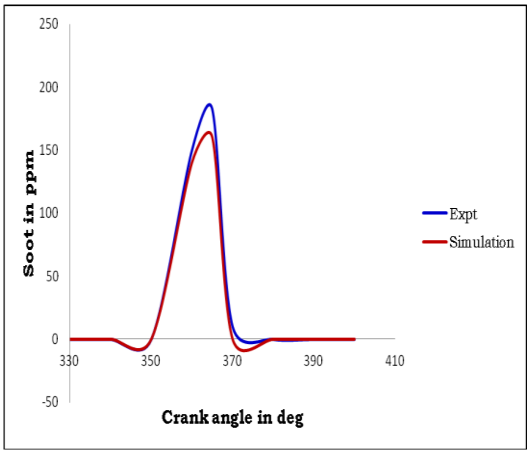


Fig.9.3 Graph: Soot curve for single Injection

9.4. Validation For Heat Release Rate

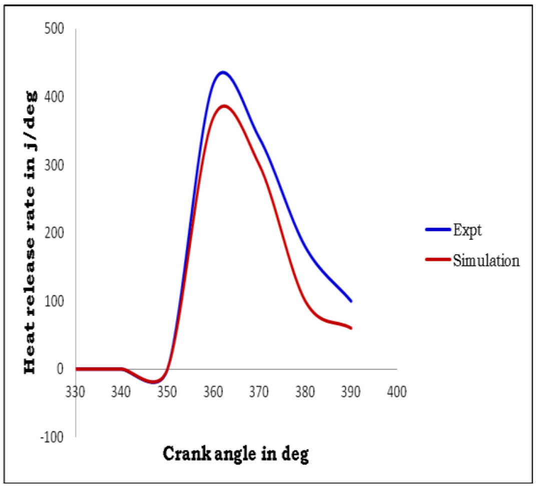


Fig.9.4 Graph: Heat release rate curve for single Injection

X. CONCLUSION

The present investigation is aimed at studying the simulation for single, using CFD code, in a direct injection compression ignition engine and results are compared with the existing experimental results. The simulation was carries for single injection for which the comparison with experimental results were not made due to non availability of existing experimental results. In simulation process 2000 particles are injected during combustion and it was observed that 1890to 1980 particles are only participating in the combustion.

REFERENCES

[1]. Shimazaki, N., Hatanaka H., Yokota, K., and Nakahira, T.,"A Study of Diesel Combustion Process Under the Condition of EGR and High Pressure Fuel Injection with Gas Sampling Method", SAE Paper 960030, 1996.

[2]. Sundoh, S., Kakekawa, T., and Tsujimurwa, K., "The Effect of Injection Parameters and Swirl on Diesel Combustion with High Pressure Fuel Injection", SAE Paper 910489, 1991.

[3]. Epsey, C. Pinson, J. A., and Litzinger, T.A., "Swirl Effects on Mixing and Flame Evolution in a Research DI Diesel Engine", SAE Paper 902076, 1990:1-10

[4]. Ogawa, H., Matsui, Y, Kimura, S., and Kawashima,J.,"Three- Dimensional Computation of the Effects of the Swirl Ratio in Direct- Injection Diesel Engines on NOX and Soot Emissions", SAE Paper 961125, 1996.

[5]. Zhang, L., Takatski, T., and Yokota, k., "An Observation and Analysis of the Combustion under Supercharging on a DI Diesel Engine", SAE Paper 940844, 1994.

[6]. Jafer, D., "Pilot Injection", Engineering. October 15, 1937.