CHAPTER – 2
REVIEW OF LITERATURE

2.1 Introduction

Over the last century, IC engines continued to develop, as our knowledge of engine processes has increased. With the advent of new technologies the demand for new types of engines has risen. Further, the environmental constraints on engines have become stringent during the recent years. Last three decades has witnessed explosive growth in engine research and development with the key issues like, market competitiveness, stringent emission norms and fuel economy. A very large amount of literature on engine research is available. In the following sections, the literature related to this work is presented.

The effect of combustion phenomenon in DI diesel engine has been highly influenced by the induction generated swirls and piston bowl configuration. Being heterogeneous in nature, controlling combustion in a diesel engine significantly increases its efficiency and pollutant formation. Better understanding on the in-cylinder fluid dynamics, fuel sprays and combustion will help in meeting stiff challenges such as fuel economy and pollutant formation. Many experimental, CFD works have been reported in the past. Critical review of the available literature pertaining to piston bowl shapes, in-cylinder flows, fuel sprays and EGR effect are presented in the following section.
2.2 Modeling of Fluid Flows in I.C. Engines

Fluid dynamics is a field of science which studies the physical laws governing the flow of fluids under various conditions. Computational fluid dynamics popularly known is used to generate flow simulations with the help of computers. CFD involves the solution of the governing laws of fluid dynamics numerically. The complex set of partial differential equations is solved in geometrical domain divided into small volumes, commonly known as a mesh or grid. CFD has enabled us to understand the world in new ways. CFD enables analysts to simulate and understand fluid flows without the help of instruments for measuring various flow variables at desired locations.

Pearson R. J. and Winterbone D. E. (1990) have described computationally efficient simulation technique, which is based on the linearised one-dimensional conservation equations for, distributed parameter systems and it is amenable to the requirements of the designer in directly assessing the comparative merits of a large number of manifold configurations. Comparisons of measured and predicted volumetric efficiency curves are presented together with predicted results. Which illustrate the benefits of variable geometry induction systems.

Takenaka and Aoyagi (1991) have used CAD/CAE procedure for the analysis of the flow in an intake port-valve-cylinder assembly of a DI Diesel engine. The flow was simulated for both, steady state conditions and transient motored situation during intake and
compression strokes. The characteristics of the helical port are analyzed, for the steady state case in terms flow rate, generation of angular momentum flux and induced in-cylinder flow motion. The results of the simulation were correlated with experimental results, consisting of oil film visualizations on valve and intake port surfaces, and of local velocity measurements in the cylinder. The transient flow simulation shows the different characteristics of the flow motion in cylinder and piston bowl during intake and compression strokes. It was observed that the swirl generating capacity of the valve is different between the valve opening and closing phases. Also, a strong interaction was found between the swirling motion and the shape and position of the piston bowl.

Nicos Lanommatos et al (1992) were describes three linked computational models which allow the estimation of swirl generated during the induction process, the modifications of swirl with bowl-in-piston combustion chambers during compression as the piston approaches TDC, the interaction of the fuel sprays with swirl including relative cross wind velocities between the air and the fuel sprays and spray impingement velocities. It is useful insights on the interaction between the fuel spray and sir, such as the magnitude of the crosswind velocity and impingement velocity when the spray impinges on the piston bowl walls.

Sweetland and Reitz (1994) have applied Particle Image Velocimetry (PIV) to a turbulent air jet and to flows in the piston bowl of a single cylinder DI diesel engine. The turbulent jet results were
used to develop methods for determining turbulence dissipation rate and length scale estimates from limited PIV data sets. These methods were then applied to the engine PIV data. Engine PIV turbulence quantities and velocity results were compared to KIVA-3 predictions. Important conclusions were 1) High and low estimates of fluctuating velocity intensities, respectively, can be obtained from limited sets of PIV data 2) The integral length scale can be estimated 3) The turbulence kinetic energy estimated from the PIV photographs agreed well with that from KIVA. 5) The integral length scales predicted by KIVA and showed good agreement with the size of the vorticity concentrations in the PIV data.

Tim Sebastian Strauss and Wolfgang (1995) have simulated a part and a full load of the production engine and a full load simulation of a modified engine design are analyzed with the SPEED CFD code. The mixing and combustion process is visualized by means of the iso-surfaces of stoichiometric mixture. The correlation of this surface with global quantities as heat release, mean pressure and temperature and swirl ratio is emphasized. The global properties are presented resolved for the swirl, main chamber and the swirl chamber throat separately. The formation of thermal NO and soot are simulated and analyzed.

Kang Y.H. Chang et al (1995) have performed in-cylinder flow simulations with the valve flow given as an inflow boundary condition from experimental measurements using KIVA-3 code. They concluded that the predicted swirl ratios in a steady flow rig and an operating
engine are comparable with each other. There are some differences in the 3-D flow structures in a steady flow rig and operating engine. An organized swirl is produced by axial development of the flow in a steady flow rig while merging of multiple swirls and tumble motion in a engine.

Andrew McLandress et al (1996) have used CFD code KIVA-3 to simulate the flow through the port and cylinder of the engine. Results reveal that mixing of the intake and residual gases is highly non-uniform. Many complex flow structures develop during intake and are destroyed during compression. The flow field near top dead center exhibits spatial in homogeneities in temperature, kinetic energy and its dissipation rate.

Bo. T et al (1997) have studied 3-D CFD calculations of the gas motion and spray characteristics of a small high speed DI diesel engine using SPEED code. The accuracy of the predictions is assessed through comparisons with detailed LDA measurements of the velocity field during the induction and compression strokes up to the time of ignition, as well as with quantitative measurements of the spray penetration and local droplet velocities. They found that the predictions of spray penetration and droplet velocities for the most part reproduce the experimental data.

Ronald and Sunil (1997) have developed a computational methodology for prediction loss in intake regions of IC engines. This methodology, which is has been successfully tested and validated against benchmark quality data for a wide variety of complex 2-D and
3-D laminar and turbulent flow situations, is applied here to a loss prediction problem from industry. Total pressure losses in the intake region of a Caterpillar diesel engine are predicted computationally and compared to experimental data. Detailed documentation is provided for the location of pockets, mechanisms responsible for losses and the relative magnitude of different loss sources. Good agreement was found between predicted and measured results.

Xavier et al (1997) have investigated on in-cylinder motion, during the intake and compression strokes. The engine is equipped several optical accesses on cylinder liner and cylinder head. The turbulence and local velocity in combustion chamber were measured by LDV. They demonstrated the influences of the bowl location on the tumble charge angular momentum at the end of the intake stroke.

Fuchs T.R. and Rutlan C.J. (1998) have simulated intake, compression and combustion of a Caterpillar diesel engine using KIVA-3 CFD code. Seven variations on intake and two injection schemes were explored. The results revealed that, in many cases one of the three factors i.e. swirl ratio, temperature and turbulence was dominated in describing a combustion or emission behavior.

Barths et al (1998) have studied comprehensive chemical mechanisms, which include all relevant chemical combustion processes that occur in a DI Diesel engine during auto-ignition, the burnout in the partially premixed phase, the transition to diffusive burning and formation of pollutants like NO, and soot. The complete structure of the flame is preserved without simplifying highly
nonlinear dependencies of the chemistry. Using the Representative Interactive Flamelet model the one-dimensional unsteady set of partial differential equations is solved online with the 3-D CFD code. Applying this model pollutant formation in acetane fueled Volkswagen DI 1900 Diesel engine is investigated. It is shown that the soot emissions are primarily controlled by the mixing process in the cylinder. Numerical simulations for different injection rates are compared to in house experiments, where cylinder pressure and the exhaust gas concentrations of NO, and soot was measured.

Chen et al (1998) have studied on engine flow using commercial CFD package STAR-C. The Volkswagen (VW) two-valve transparent 1.9 l DI diesel engine was used to simulate flow field in the inlet port and combustion chamber under engine speeds of 1000 rpm and 2000 rpm. The CFD predictions were verified by Laser-Doppler anemometer measurements under motored condition during three periods: valve opening, valve closure and the maximum valve lift. In presentation they discussed the predicted flow structure and its main features under different engine speeds, and examined the validity and accuracy of the CFD predictions in comparison with measured data.

Bianchi et al (1999) have investigated influence of different initial condition definition procedures on combustion and emissions predictions in small-bore high-speed D.I. diesel engines. Using the STAR-C code for the intake stroke calculation and KIVA-II for the compression stroke and combustion simulation performed the analysis. The study considered a detailed assignment of the initial
conditions at IVC. In the second case the results of a time-resolved calculation of the intake stroke performed using the STAR-CD code was used. An algorithm was developed in order to properly map the data from STAR-CD into KIVA-II.

Tadao Okazaki et al (1999) have developed simulation techniques, which play important role on contemporary engine design. In his study, CFD approach was focused to design the intake and combustion system of the direct injection diesel engine for versatile use. A practicality was stressed as much as an accuracy to correspond to designer and researcher’s requirements, such as close relationship to the engine performance and short period of computation. The correlation of the trapping efficiency and the swirl ratio was mainly focused. A steady flow rig tests and engine operation data were combined to improve their quality mutually.

Liu Shenghua et al (1999) have developed a new type of swirl generator in a short straight port to get induction swirl in the cylinder. The generator consists of several curvilinear diversion blades and is installed near to the inlet valve. Engine tests proved that variably induced swirl has a significant effect on diesel engine fuel consumption.

Chiu H.H. (2000) has summarized the major theoretical accomplishments in droplets and sprays in the twentieth century in his two-part article, with emphasis focused upon the evolution of scientific concepts, paradigms and methodologies. A structural spray theory, which was developed form an early view of isolated droplets,
has evolved into a new view that the interaction droplet and micro-scale structures and clusters of many-systems are also fundamental entities in practical sprays. Outstanding issues and critical bottlenecks that have prevented further advancement of the existing analytical theory of droplet physics are examined, and an emerging research trend in a unified theory of droplet phenomena discussed. Recent accomplishments and future prospects of a unified theory are presented to coupling the status of this special branch of droplet science and its future application.

Randall and William (2005) have studied the quasi-steady vaporization and combustion of multiple-droplet arrays is studied numerically. Vaporization rates, flame surface shapes, and flame locations are found for different droplet array configurations and fuels. The number of droplets, the droplet arrangement within the arrays, and the droplet spacing within the arrays are varied to determine the effects of these parameters. As a result of the droplet interactions, the number of droplets and relative droplet spacing significantly affect the vaporization rate of individual droplets within the array, and consequently the flame shape and location. For small droplet spacing, the individual droplet vaporization rate decreases below that obtained for an isolated droplet by several orders of magnitude. Similarity parameter, which correlates vaporization rates with array size and spacing, is identified. Individual droplet flames, internal group combustion, and external group combustion can be observed depending on the droplet geometry and boundary conditions.
Anand Kumar T. et al (2005) have used a KIVA code to analyze the in-cylinder flow field of four stroke gas engine. In their study simulation results were compared with available experimental results. The effect of different motoring speeds on the intake-generated turbulence, mass flow rate, velocity, swirl ratio and TKE were analyzed.

Alfred and Ganesan (2005) have studied the effect of cylinder swirl patterns along the cylinder axis obtained during early compression, on bowl flow velocities and generated TKE at TDC. They concluded that the swirl magnitudes produced in the piston bowl at TDC are strongly influenced by the swirl magnitudes persisting near the cylinder bottom or piston face during early compression.

Saraswat and Sudhakara Reddy (2005) present detail spray dynamics, the various spray combustion and sub-system models. The concepts like droplet group effects, droplet dispersion and group combustion are discussed. The status in the application of spray combustion models to diesel combustion, gas turbine combustion and rocket combustion are discussed.

Kuleshov (2005) has developed a model for multi-zone diesel sprays and combustion. The results allow describing the phenomenon of increased fuel consumption with increase of swirl ratio over the optimum value. The model has been used for simulation of different engines performances.

Ravindra Aglave (2007) has used CFD code KIVA III is used and modified k-ε model for describing in-cylinder turbulence. The spray
dynamics are handled using a discrete droplet model along with sub-models for collision, breakup and evaporation. He established the suitability of intrinsic low-dimensional manifold in simulating turbulence-chemistry interactions using a presumed PDF approach with greater accuracy in predicting kinetically controlled process.

Vijay Manikandan et al (2007) have studied on engine emissions by establishing a method of prediction of concentration various species in the CI engine exhaust based upon the instantaneous pressure rise in the combustion chamber. Accurate predictions are achieved for all particulate and gaseous emissions except for Hydrocarbon emissions.

Piston Bowl Configurations

Mehta et al (1993) have investigated on the matching of fuel injection characteristics with air motion and combustion chamber geometry of DI diesel engines to obtain improvements in fuel economy and emission characteristics. The spray development aspects are also briefly discussed. Different analytical approaches to model air entrainment in turbulent jet in the engine situation are summarized. Critical aspects of spray-swirl interaction and the important features of different simulation models are discussed.

Too D. Fansle (1993) have conducted experiments on turbulence and its effect on improved fuel-air mixing and combustion in DI engines incorporate high-squish piston bowls and intake-generated air
swirl. They demonstrated the inflorescences of the bowl location on the tumble charge angular momentum at the end of intake stroke.

Long Zhang et al (1995) have conducted an experimental study aiming to investigate the effects of combustion chamber geometry on combustion process in an optically accessible DI diesel engine. The combustion processes of three different chamber geometries, included the production type, were revealed and the flame movement behaviors such as the distribution of flame velocity vectors and the averaged flame velocity inside and outside the combustion chamber were measured by means of a cross-correlation method. By comparing the flame movement and distribution between different chambers and nozzle protrusions, the results showed that the chamber geometry has significant effect on the flame velocity, the flame velocities of the reentrant chamber were larger than that of the dish chamber during expansion period. The flame distribution inside and outside the chamber is considerably affected by the chamber wall geometry, and the reentrant chamber prevents the flame from spreading over the squish region. The relationship between these changes of the flame behaviors and its smoke emissions measured in the same engine under the same experimental conditions are also discussed.

Senecal et al (1997) have studied 3-D CFD code, based on the KIVA code, is used to explore alternatives to conventional DI diesel engine designs for reducing NOx and soot emissions without sacrificing engine performance. The effects of combustion chamber design and fuel spray orientation are investigated using a new
proposed GAMMA engine concept and two new multiple injector combustion system designs, which utilize multiple injectors to increase gas, motion and enhance fuel/air mixing in the combustion chamber. From these computational studies, it is found that both soot and nitrous oxide emissions can be significantly reduced without the need for more conventional emission control strategies such as EGR or ultra high injection pressure.

Bruno Dillies et al (1997) have enhanced models of initial version of the KIVA-2 code to allow computation of quite complex geometries with different physical sub-models for turbulence, injection and combustion. The spray model was also modified in order to limitation the deficiencies of current spray models when computing high injection pressure Diesel sprays. Numerical results were compared to experimental data from two different DI Diesel engines. First of all LDV measurements were performed in the bowl of an optically accessible engine. It was found that the predicted mean and turbulent velocities are in good agreement with experimental data. Calculated velocities are slightly under predicted but the computations show the same trends as the experiments with engine RPM. The combustion engine database includes pressure profiles and combustion analysis in a second engine for a wide range of engine parameters such as RPM, load, injection timing. Provided that the injection conditions are precisely specified, a fair agreement between experimental and numerical results was obtained.
Zolver et al (1997) have carried out multidimensional computations to aid in the development of a direct injection Diesel engine. Intake, compression, injection and combustion processes are calculated for a turbo-charged direct injection Diesel engine with a single intake valve. The effects of engine speed and engine load, as well as the influence of EGR are compared to experimental measurements. The influence of piston bowl shape is investigated. 3-D calculations are performed using a mesh built from the complete CAD definition of the engine, intake port, and cylinder and piston bowl. The injection characteristics are found to be of primary importance in the control of the combustion process.

Beard et al (1998) have performed simulations using KMB code, a modified version of KIVA-II along with the detailed flow field description integral quantities characterizing the flow are derived. In results it is presented that the role of the re-entrant in the W-shaped bowls to increase both swirl number and turbulence level around TDC.

George Papageorgakis et al (1998) have adopted RNG model to perform computations in realistic engine geometries. The parameters explored include the effects of piston crown shape, injector targeting, glow-plug presence, injection velocity, injection timing, number of injector holes and initial swirl ratio on mixing. It was concluded that the Mexican-hat design for piston enhances mixing due to the tumbling motion of the mixture caused by that particular piston
shape. The optimum injector hole angle was found to be $45^\circ$ with respect to the horizontal.

Yoshiyuki Kidoguchi et al (1999) have carried out experiments to reduce $NO_x$ and particulate emissions simultaneously in a DI diesel engine based on the concept of two-stage combustion. At initial combustion stage, $NO_x$ emission is reduced with fuel rich combustion. At diffusion combustion stage, particulate emission is reduced with high turbulence combustion. The high squish combustion chamber with reduced throat diameter is used to realize two-stage combustion. This combustion chamber is designed to produce strong squish that causes high turbulence. When throat diameter of the high squish combustion chamber is reduced to some extent, simultaneous reduction of $NO_x$ and particulate emissions is achieved with less deterioration of fuel consumption at retarded injection timing. Further reduction of $NO_x$ emission is realized by reducing the cavity volume of the high squish combustion chamber.

Bianchi and Pelloni (1999) have performed numerical analysis aimed to develop a combustion chamber suited for small bore D.I. Diesel engine equipped with Common-Rail injection systems. The CFD methodology has been validated through an extensive comparison with measurements of mass production engine for passenger car. In order to reduce spray wall impingement a new larger diameter bowl has been designed and compared to the traditional low-diameter-bowl of the experimental engine.
Hideyuki Ogawa et al (2000) have analyzed the fuel spray distribution in a DI diesel engine was actively controlled by pilot and main fuel injections at different piston positions to prevent the main fuel injection from hitting the pilot flame. CFD analysis demonstrated that the movement of the piston with a cavity divided by a central lip along the center of the sidewall effectively separates the cores of the pilot and main fuel sprays. Experiments showed that an ordinary cavity without the central lip emitted more smoke, while smokeless, low NOx operation was realized with a cavity divided by a central lip even at heavy loads where ordinary operation without pilot injection emits smoke.

Rahman et al (2000) have studied on reductions in fuel consumption and pollutant emissions from direct injection diesel engines. In their studies, improvement in mixture formation was attempted by changing the combustion chamber geometry. The fuel spray development was visualized from two directions in an actual engine with a transparent cylinder and piston arrangement. The place where spray impinges, the distance from impinging wall to nozzle tip, and the shape of the chamber entrance and bottom were varied to determine, their effects on the fuel spray development in the combustion chamber. They found that a re-entrant type combustion chamber with round lip and round bottom corners provides better air and fuel distribution than a simple cylindrical combustion chamber. They proposed modifications of the available empirical equations for
calculating the spray path length and the area where spray spreads were attempted.

Song-charng et al (2003) have investigated the effects of flow turbulence on premixed iso-octane HCCI engine combustion. Using different piston geometries, namely a disc-shape versus a square-shape bowl, generates different levels of in-cylinder turbulence. The results are presented on the effect of turbulence on wall heat transfer and estimate of initial mixture conditions on combustion. It is found that combustion and emission are very sensitive to the initial mixture temperature. The square-bowl piston generates higher turbulence levels and results in higher wall heat flux and longer combustion durations.

Wickman (2003) has performed on combustion chamber geometry design optimization on a HSDI diesel engine using KIVA-GA code. The optimum split-spray piston design was demonstrated experimentally to be able to achieve low emissions when using retarded injection timing, high swirl ratio, high injection pressure, high boost pressure, high EGR rate, and a low compression ratio.

Fontanesi et al (2005) have studied on mixture formation within the combustion chamber on HDDI diesel engine for marine applications. They concluded that that the modifications in the combustion chamber shape proved to influence the spray evolution within the combustion chamber, due to increased swirl intensity and to higher turbulence levels the EVO piston shows improved air usage and better air/fuel mixing. The benefits brought in by the piston
modifications are confirmed by a final combustion simulation, showing that the EVO piston leads to both an increase of global engine performances and a small but significant reduction in terms of pollutant formation.

Salvador et al (2005) have applied multi-zone chemical kinetics model to analyze HCCI for two combustion chamber geometries with different levels of turbulence. Results indicated that the high turbulence square geometry has longer burn duration than the low turbulence disc geometry.

Andreas et al (2007) have studied how geometry generated turbulence affects the rate of heat release in an HCCI engine. Two different combustion chamber geometries were investigated, one with a disc shape and one with a square bowl in piston. Chemiluminescence imaging showed that the combustion starts in the bowl and propagates into the squish volume and the combustion started in the same corner of the bowl in all cycles. Further it is showed that the combustion was more stratified in the bowl case probably due to temperature in-homogeneities.

### 2.3 Fuel Sprays and Injection Strategies

It is found that the fuel spray plays an important role in exhaust emissions. From the recent trends, some of the most significant improvements in the diesel engine emissions have resulted in various injection strategies like injection timings and split injections. In the literature, it has been discussed indicating that
multiple-injection is a powerful tool, which can help in achieving the reduced emissions required for future emission norms. In industry, the current strategy for particulate reduction is to use high injection pressures with small hole nozzles in the injector tip. Critical review of the available literature pertaining to the experimental and CFD works has been reported for various injection strategies and nozzles presented in the following section.

David J. Timoney and J. Smith (1996) have conducted experiment on a 121mm bore, single-cylinder, deep-bowl, non-swirling D.I. diesel research engine. Heat release and instantaneous injection rate data was obtained from a series of experiments on, using a variety of fuel injection pump builds. Results from tests at constant air-fuel ratio and constant start of combustion angle show that increasing the mean fuel injection kinetic energy at a given engine speed reduces the heat release time and increases the fuel-air mixing rate. Also, at constant fuel injection kinetic energy, increasing the engine speed increases the fuel-air mixing rate. These experimental trends have been interpreted with the aid of a novel but mathematically very simple analytical approach, based on the hypothesis that all fuel-air mixing in a DI diesel combustion system is promoted by kinetic energy inputs. They suggest engine speed-related air turbulence effects can enhance that fuel-air mixing in a DI diesel even in so-called ‘quiescent’ combustion systems.

Lee (1997) has conducted experimental studies on spray tip penetration and dispersion in high-pressure diesel engine. Results
shown that the spray tip penetration is qualitatively different between low and high-pressure injections. For high-pressure injection, good agreement is achieved between the experimental results.

Zhilyu et al (1997) have studied air-fuel mixing in a DI spark-ignition engine through numerical models. Intake generated flow structures in engine configuration, the spray behavior and the spatial and temporal evolution of fuel liquid and vapor phases are characterized. It was found that the intake-generated flow interacts with the injected spray drops, which strongly influence fuel distributions and air-fuel mixing.

Poul Rodatz et al (1988) have utilized a KIVA-based CFD tool to simulate the effect of a common-rail injection system applied to a large, uniflow-scavenged, two-stroke diesel engine. In particular, predictions for variations of injection pressure and injection duration have been validated with experimental data. The computational models have been evaluated according to their predictive capabilities of the combustion behavior reflected by the pressure and heat release rate history and the effects on nitric oxide formation and wall temperature trends.

Bianchi and Pelloni (2000) in their research focused on high-pressure-driven diesel fuel spray simulation and a hybrid model. In order to take into account the flow conditions inside the nozzle on liquid jet breakup, the atomization has been simulated, by using the model proposed by Gosman. Different values of the atomization model constant have been tested, the best agreement with measurements
has been found. The extensive comparison between numerical results and experiments has shown the reasonably good accuracy of the hybrid model in predicting the breakup process of high-density high-pressure-driven sprays. An isotropic model of the droplet turbulent dispersion in conjunction with the k-ε model has been presented. Significant changes in the spray structure and droplet size were detected, by comparing the isotropic and the anisotropic approaches, especially near the nozzle exit.

Andreas et al (2000) have simulated using KIVA code, the spray targeting, by way of enhanced secondary atomization was found to be a powerful way of enhancing cold start. Optimal spray targeting for cold-start performance may lead to deteriorated performance at other operating conditions.

McCracken et al (2001) have studied on the interaction of the swirl with sprays in a Diesel engine through a computational study. A multidimensional model for flows, sprays, and combustion in engines is employed. Results from computations are reported with varying levels of swirl and initial turbulence in two typical Diesel engine geometries. It is shown that there is an optimal level of swirl for each geometry that results from a balance between increased jet surface area and, hence, mixing rates and utilization of air in the chamber. Results are also reported for several split injection strategies to assess whether the interactions observed with a single injection event are applicable.
**Injection Strategies**

Gunner et al (2001) have investigated on possible benefits of split injections. The simulation results suggest that both fuel economy and NOx emissions can be improved when a two-pulse injection is used such that the late injected, stratified fuel ensures stable injection is used such that the late injected, stratified fuel ensures stable ignition at the spark plug whereas the early injected, diluted fuel reduces the maximum combustion temperature and controls NOx formation.

Andrea Bertola et al (2001) have analyzes the influence of fuel properties and injection parameters on the particulate number size distribution. They indicated that the injection parameter settings have the predominant role in the shape of the particulate matter size distribution. The total particle number detected in the exhaust gas is very sensitive to the variation of the injection pressure. Post-injection lowers the particle number. EGR shifts the maximum in the number size distribution to more and larger particles.

Taewon Lee et al (2003) have studied on how to overcome the trade-off between NOx and particulate emissions for future diesel vehicles and engines and it is necessary to seek methods to lower pollutant emissions. This study demonstrates the emission reduction capability of split injections, EGR, and other parameters on a HSDI diesel engine equipped with a common rail injection system using an Response Surface Method (RSM) optimization method. The optimizations were conducted at 1757 rev/min, 45% load. Six factors
were considered for the optimization, namely the EGR rate, Start of Injection, intake boost pressure, and injection pressure, the percentage of fuel in the first injection, and the dwell between injections. In addition, the effects of swirl were investigated. The benefit of the combined effects of EGR with supercharging was well realized by the RSM optimization, which resulted in simultaneous reductions in both NOx and PM emissions, while even improving brake specific fuel consumption. In addition, observation of the optimization process provides a more thorough understanding of HSDI diesel combustion since the observed trends can be explained by monitoring the effects of the operating parameters.

Inderpal Singh et al (2003) have investigated on the combustion and emission characteristics of a high speed, small-bore, DI, single cylinder, diesel engine using two different nozzles. The experiments were conducted at conditions that represent a key point in the operation of a diesel engine in an electric hybrid vehicle. The experiments covered fuel injection pressures ranging from 400 to 1000 bar and EGR ratios ranging from 0 to 50%. The effects of nozzle hole geometry on the ignition delay, apparent rate of energy release, NOx, Bosch smoke unit, CO and HC are investigated. The results show that the 430 VCO produced longer ignition delays and cool flames of longer durations and higher intensities than its counterpart, the 320 mini sac nozzle, under all the operating conditions at zero EGR, the analysis of the data for the two nozzles showed the dependence of NOx on the premixed combustion fraction, and the dependence of smoke
on the mixing controlled and diffusion controlled combustion fraction. The impact of EGR on this dependence is examined. Also, the trade-off between the engine-out emissions of NOx and Bosch smoke unit is determined for the two nozzles.

Cheolwoong et al (2004) have studied the effect of pilot, post and multiple-fuel injection strategies on engine performance and emissions. The study was carried out on a single cylinder optical direct injection diesel engine equipped with a high-pressure common rail fuel injection system. Spray and combustion evolutions were visualized through a high-speed charge coupled device camera. Tests were performed at fixed engine speed with various injection and flow parameters. Engine performance, emissions and cylinder pressure were obtained and analyzed to investigate the effect of fuel injection and engine operations. They found that pilot-injection reduced the ignition delay of main injection and then contributed to the improvement of power output by controlling the premixed combustion. The post-injection was found to be very effective in completing the oxidation process and reducing the particulate emissions even when small fuel quantities were injected. It was also found that multiple injection could reduce particulate emissions by more than 40% in some cases.

Ra and Reitz (2005) have studied with the numerical analysis on the effects of injection parameters and operating conditions for diesel fuel HCCI operation with consideration of Variable Geometry Sprays (VGS). Two-pulse, hollow-cone sprays were also simulated. The
results show that VGS is effective in minimizing wall wetting and allows wall wetting to be decoupled from ignition timing control. The results indicate that VGS is a promising methodology to be used to control diesel fuelled HCCI engine operation.

Gill et al (2005) have carried out experimental study on the multiple fuel injection process and its effect on the mixing and combustion in a single cylinder diesel engine with optical access. The engine is equipped with a production type cylinder head and a high-pressure common rail fuel system, which comprises a directly driven high-pressure fuel pump, and a control system capable of 8 injections per stroke. The in-cylinder spray and combustion were visualized at 10,000 fps by a high-speed color video camera and a copper vapor laser. The high-speed video recordings and in-cylinder pressure and heat release analysis for up to four fuel injections will be presented and discussed. Two pilot injections resulted in increased injection delay over a single pilot. Heat release, cylinder pressure and IMEP of 800 and 1000 bar were reduced, while that of 1200 bar as substantially increased. Post injection strategies can be beneficial in reducing peak pressure while increasing IMEP.

Manshik Kim et al (2006) have performed to investigate the combustion process in the premixed compression ignition regime in a light-duty diesel engine. As the injection timings were advanced from 15°bTDC to 51°bTDC, distinct regions of emission characteristics controlled by mixing and the combustion bowl geometry were explained. They predicted that the carbon monoxide emission and the
pressure history for early injections depend critically on key oxidation reaction rates in the chemical mechanism.

Hardy and Reitz (2006) have developed a micro-genetic algorithm was utilized to optimize a hybrid, double-injection strategy, which incorporated an early, premixed pilot injection with a late main injection. The fuel injection parameters, intake boost pressure, and EGR were considered in the optimization.

Kulshov A.S. (2006) has used DIESEL-RK software to simulate multiple injection strategy for full cycle thermodynamic engine. In the model the fuel spray is divided into a number of zones with different evaporation conditions. Predictions of spray tip penetration, spray angle and ignition delay were validated by the published data. And formulas for computation of these characteristics were derived.

Michael J. Bergin et al (2007) have conducted experiments with high-speed video imaging in a swirl-supported DI diesel engine. It is found that without swirl, combustion-generated gas flows influence mixing on both sides of the jet equally. In the presence of swirl, the heat release occurs on the leeward side of the fuel sprays, this leads to lower local equivalence ratio and lower soot production rates with swirl.

Jesus Benajes et al (2008) have analyzed the potential of the highly premixed combustion concept for pollutant control in a HD diesel engine. A narrow angle nozzle configuration is investigated with two different adapted piston bowl geometries. The results showed that NOx emissions are lower for narrow angle configurations due to lower
flame temperatures attained during combustion. Open bowl design offers better emission levels and engine efficiency at medium load than the straight wall bowl design. Roughness of the combustion due to the high amount of fuel burnt in premixed but non-homogenous conditions leads to important drawbacks in term of noise and engine life duration. The liquid fuel film formed on piston surface burns in poor air/fuel mixing conditions, producing high levels of soot. Moreover, the non-optimized interaction between the pilot premixed and the main diffusion-controlled combustion stages also contributes to increase soot emissions.

### 2.4 EGR and Emission Reduction

Exhaust gas re-circulation is a NOx emission reduction technique used in internal combustion engines. EGR dilutes the in-cylinder fluids by intermixing the re-circulated exhaust gas with incoming air. However, exhaust gas acts as an inert gas with higher specific heat than air, which lowers the peak combustion temperatures and thereby reducing the NOx formations. EGR also lowers the adiabatic flame temperature, the maximum temperature that can be achieved for given reactants. Experimental and CFD works have been reported in the EGR. Critical review of the available literature pertaining to EGR is presented in the following section.

Kazuhiro Akihama et al (1995) have demonstrated the smokeless rich diesel combustion. This can realize smokeless and NOx-less combustion by using a large amount of cooled EGR under a
near stoichiometric and even in a rich operating condition. They focused on the effects of reducing diesel combustion temperature on soot reduction. In their work the smoke suppression mechanism in the smokeless rich combustion, where the temperature is reduced by higher EGR rate, is analyzed.

Pitsch et al (1996) in their study of numerical simulations for a DI Diesel engine with EGR are performed in order to predict the chemical details of the combustion process and the resulting pollutant formation. The gas phase chemistry has been described with a detailed chemical mechanism, which accounts for ignition, the oxidation of the fuel, \( \text{NOx} \) formation and destruction, and the build-up of poly-cyclic aromatic hydrocarbons (PAHs) up to four aromatic rings. The further growth of the PAHs, the formation of soot particles, and the interaction of particles with other particles and the gas phase are described by a kinetically based soot model. \( \text{NOx} \) formations are described by thermal, prompt, nitrous, and re-burn chemistry. Comparisons of the calculations with experimental data for pressure, ignition delay, soot mass, \( \text{NOx} \), and major species concentrations show good agreement. The influence of the EGR rate on soot and \( \text{NOx} \) formation is investigated and discussed.

Gosta Liljenfeldt et al (1997) have simulated the combustion process and \( \text{NOx} \) emission of the world’s largest medium speed diesel engine using KIVA-2 code. The simulation results, such as the cylinder pressure, heat release rate were compared to the measured
values. It is shown that the modified KIVA-2 combustion simulation
tool works quite correctly.

Doo Sung Baik et al (2001) have studied the combustion and
reduction characteristics of NO, O₂, and CO₂ on the 11,000cc
turbocharged diesel engine equipped with an EGR valve. Also, tried to
make a selection of optimized EGR rates based on these results it is
found that the emission levels of CO₂ were substantially increased
according to the increase in EGR rate. The reduction rate of NO
increased at the higher load, rather than lower load.

Bryzik et al (2003) have studied on integrated control strategies
for the Optimization of injection Pressure, EGR ratio, injection Retard
or Advance and Swirl ratio (O.P.E.R.A.S) are demonstrated. The
strategies are based on an investigation of combustion and emissions
in a small bore, high speed, DI diesel engine. The engine is equipped
with a common rail injection system and is tested under simulated
turbocharged engine conditions at two loads and speeds that
represent two key operating points in a medium size HEV vehicle. A
new phenomenological model is developed for the fuel distribution in
the combustion chamber and the fractions that are injected prior to
the development of the flame, injected in the flame or deposited on the
walls. The investigation covered the effect of the different operating
parameters on the fuel distribution, combustion and engine-out
emissions. Illustrations of integrated strategies for O.P.E.R.A.S., that
take into consideration the needs of the after treatment devices, are
given.
Stuart W. Neill and Chippior (2003) have studied the exhaust emissions from a single-cylinder version of a heavy-duty diesel engine with EGR using 12 diesel fuels derived from oil sands and conventional sources. The test fuels were blended from 22 refinery streams to produce four fuels at three different total aromatic levels (10, 20, and 30% by mass). The cetane numbers were held constant at 43. Exhaust emissions were measured using the AVL eight-mode steady-state test procedure. PM emissions were accurately modeled by a single regression equation with two predictors, total aromatics and sulphur content. Sulphate emissions were found to be independent of the type of sulphur compound in the fuel. NOx emissions were accurately modeled by a single regression equation with total aromatics and density as predictor variables. PM and NOx emissions were significantly affected by fuel properties, but crude oil source did not play a role.

Zhu et al (2003) have used such techniques that reductions in fuel consumption, noise level, and pollutant emissions such as, NOx and PM, from DI diesel engines. To achieve these reductions, many technologies such as high injection pressure, multiple injection, retarded injection timing, EGR, and high swirl ratio have been used in high-efficiency DI diesel engines in order to achieve combustion and emission control. They present a computational study of both the individual effect and their interactions of injection timing, EGR and swirl ratio separately and their interaction in a HSDI common rail diesel engine using the KIVA-3V code. It was found that the
calculated results agree well with the experimental data obtained and there exists a set of optimal injection timing, EGR and swirl ratio for simultaneous reduction in both NOx and soot.

Morgan R.E. et al (2003) have worked on a rapid combustion machine at ambient conditions representative of a modern high speed DI diesel engine. The qualitative soot concentration was determined using the laser induced incandescence technique. They found that peak soot concentration values were found to correlate well with the velocity of the injected fuel jet. Change air pressure was observed to have minimal effect on the peak soot concentration.

Jeffrey et al (2004) manufacturers of heavy-duty diesel engines for sale in the United States, focuses on the technology solutions possible for engine makers for the interim 2007-2009 timeframe and discusses the additional NOx reduction strategies for a 2010 compliant engine. The possibility of achieving a larger portion of the interim 2007-2009 NOx standard through in-cylinder control methods rather than by NOx exhaust treatment is discussed. High levels of EGR and advanced injection strategies to modify the conventional diesel combustion process are just two processes that can be accommodated in many of today’s engine designs. Variable valve actuation will help control air management and residual as well as provide exhaust gas temperature control for exhaust treatment synergy. Advanced control strategies will be used to control transient emissions. Diesel particulate filters will be used for the primary particulate control as of 2007, notwithstanding additional PM
countermeasures that can be obtained from fuel system control. The remaining significant issues of fuel consumption and durability are discussed in light of alternative solutions.

Schwoerer et al (2004) have studied the Internal EGR with retarded injection timing can provide a 30% reduction in diesel NO\textsubscript{x} emissions and is an attractive solution to meeting NO\textsubscript{x} emission levels for heavy-duty diesel engines, especially for off-road and vocational applications. At lower NO\textsubscript{x} emissions levels, IEGR may be used to supplement cooled EGR or to control HCCI combustion. Alternative valve actuation strategies for IEGR are reviewed. A valve actuation system to provide on-off control of IEGR combined with compression release braking is presented. System design and simulation results are reviewed. Engine performance predictions and initial test data are discussed, including turbocharger sizing and particulate emission considerations.

Ryoji Nishiumi et al (2004) have investigated the effects of cetane number and distillation characteristics of paraffinic diesel fuels on PM emission in order to clarify optimum FTD fuel properties for conventional DI diesel engines with turbocharger, cooled EGR and common rail system.

Mark P.B. Musculus (2004) is measured exhaust NO and NO\textsubscript{x} emissions and correlated with observations of soot luminosity and jet penetration as the intake-temperature and injection timings were varied. He noticed that injection timing was retarded to the misfire limit, dramatic reductions in soot luminosity accompanied increased
exhaust NOx emissions. Thermal NO formation increases with temperature. The reduction of the cooling effect of soot radiative heat transfer may increase the actual flame temperature and thus NO formation.

Stephen (2005) has described the modern HD diesel engine, as used in on-highway applications, and discussed the technologies that make these engines possible. The evolutions of emissions requirements worldwide are discussed. He discussed about the advanced technologies that will be candidates for meeting future stringent emission standards.

Aravind et al (2005) experimented by using Di Butyl Melate blend ratios. There is an improvement in smoke, particulate matter, THC and improvement in thermal efficiency. They concluded that by using the blends of diesel and Di Butyle maleate the emissions were reduced and simultaneous improvement in the brake thermal efficiency. The reduction in emission was due to increase of oxygen content in fuel. Due to complete combustion the peak temperature is increased and hence formation of NOx is more.

Mark P.B. Musculus (2005) has studied the influence of NO formation of local cooling due to soot radiative heat transfer in DI diesel engine. The contribution of burned-gas compression-heating of NOx formation was also estimated through thermodynamic analysis. He concluded that radiative cooling normally present in diesel combustion reduces flame temperatures and an estimated NOx
reduction of 12-25%. In the limit of slow mixing, compression heating of burned gases can increase flame temperature.

Cloddna et al (2005) have focused on the formulation and subsequent assessment of a semi-empirical calculation scheme, which estimates the $NO_x$ concentration found in the exhaust stream of a CI engine. The semi-empirical modeling approach investigated has revealed itself as a potentially useful prediction tool, which could find application, as part of an on-board fast response $NO_x$ sensing system used in conjunction with exhaust after-treatment systems.

Hyungsuk and Patrick (2005) have conducted the experiments by increasing torque at constant speed and by increasing speed at constant torque, in conventional diesel combustion regions. The emissions from the two locations are compared. The transient effects of EGR rates and injection timing on HC and $NO_x$ are described and the effects of linear and step load change on emissions are compared. Base EGR was set with an EGR valve on the engine the position of the valve was then held constant throughout the transient. The pressure history in one of the cylinders was measured as well. The results show that peak emissions of $NO_x$ occurred when a step load change was applied for constant speed and constant torque cases at low EGR levels, but there were no evident step load change effects on $NO_x$ when higher EGR levels were used. $NO_x$ emissions were very sensitive to transient operation of changing injection timing and EGR rates for constant speed and constant torque cases. The HC measurement showed little sensitivity to transients when the EGR valve was closed.
The HC emission sensitivity to transients increased dramatically when EGR rate was increased. It is noted that injection timing and EGR rate affected engine torque and speed for a constant quantity of fuel injected.

Ravindra Mahjan et al (2007) have conducted experiments on flexibility in injection parameters and EGR to optimize emission and noise in DI diesel engine. They conclude that wider bowl with high injection pressure results in better air-fuel mixture. With wider bowl, nozzle tip protrusion needs to be fine tuned for reduced smoke and BSFC results at full load.

2.5 Summary

The review of literature describes the progress made in the field of modeling of in-cylinder flows of diesel engine applications. It is clear that the research in this direction started early in 1980’s with the development of 2-D models to simulate in-cylinder flows and combustion of IC engines. Many satisfactory results for in-cylinder flows with fuel sprays have been reported. With these models the gas flow pattern were predicted with reasonable accuracy but the prediction of combustion is posing a problem.

The progress in the 3-D modeling is witnessed from the 90’s, with the availability of high-speed computers. Accurate prediction of engine process such as in-cylinder flows, sprays, combustion and pollutant formation is possible but most of the 3-D calculations are reported on super computers and parallel processing based machines.
The development in PC technologies is encouraging to take up these complex 3-D modeling on a personal computer.

Many researches are trying to model in-cylinder flows with the modifications of inlet manifolds, valve profiles and piston bowl shapes. With the review of literature it is observed that piston bowl shape is highly influencing on compression swirls eg. Yoshiyuki Kidoguchi (1999) demonstrated that high squish combustion chamber produce strong squish flows that causes high turbulences. Ogawa (2000) demonstrated that piston which is divided by a central lip effectively reduces $NO_x$ emissions. Hence, it is interesting to compare squish lip bowl with most common bowls like HSB and TB.

After introducing common rail injection system in DI diesel engines, many researchers have tried and succeeded to improve the engine performance with the various injection strategies. Hence, it is interesting on pilot-injection.

After imposing emission norms, many researchers are trying to reduce $NO_x$ emissions with the inert gas recalculations and succeeded. This has motivated to work in this direction.