International Journal of Engineering and Advanced Technology (IJEAT)
ISSN: 2249-8958 (Online), Volume-9 Issue-3, February 2020

Emission Characteristics of Single Cylinder CI Engine for Varying Nozzle Depth at Different Spray Angles

Rahul Kumar Singh, Dinesh Kumar Soni

Abstract: *The present research is about the reduction of harmful gases from the exhaust of a diesel fuel operated diesel engine. A kirloskar single cylinder diesel engine (model –TV-1) is selected for the numerical simulation using commercially available software AVL FIRE. The operating and boundary condition of single cylinder diesel engine is referenced in the published articles. The hemispherical bowl piston geometry is generated and meshed in the software the research is further processed with the selection of three different spray angles like 120º, 140º, and 160º. Whereas 120 spray angle is the standard spray angle. Additionally to analyzed the effect of nozzle depth on the emission, combustion and performance parameters four nozzle depth values are also included in the research. The four nozzle depths are 0.5mm, 1mm, 1.5mm and 2mm. whereas 1mm nozzle depth is standard nozzle depth. All four nozzle depths are examined under 120º, 140º, and 160º spray angle. The need of petroleum fuels is increasing everyday despite of their speculation regarding the depletion of petroleum fuels. The petroleum fuels are categorized as suspicious element for human being due to its well known major harmful effects on the environment. The concern is that the need of petroleum fuel is increasing simultaneously with its price in the market. The researchers around the world are focusing either on the reduction of harmful effects which produced from the petroleum fuel or finding another a batter substitute of the petroleum fuel.*

Keywords: *spray angle, nozzle depth, numerical simulation, emission parameters, combustion parameters, and performance parameters.*

I. INTRODUCTION

The modern diesel engine development is governed by maximum efficiency and low emission requirements the variation in spray angle at different nozzle depth has enhanced engine efficiency and decreased emissions by improving the atomization characteristics of the fuel spray. The variation in spray angle at different nozzle depth has been studied by several researchers with different angle and different nozzle depth. The experimental investigation of diesel sprays angle and different nozzle depth is very challenging due to no any modification is possible in engine from its standard condition and position of nozzle depth inside the combustion chamber and spray angle of nozzle. Hence here we use suitable software for getting result data from this variation in engine operating condition.

Emissions from diesel engine are not only harmful for the present, but it also unfavorable for the future of the world. It must be control today by considering its hazardous effect on tomorrow. The severity of the emission problem has judged by a life cycle assessment of new diesel engine, and the results clearly show that, the global warming and photochemical ozone formation have major impact of 19.47% and 17.54% respectively, of total impacts of the diesel engine. On this stage, numerous researches are proceeding to control emissions from diesel engines by means of different facets of applied science. The technology has been used with the new dimension of nozzle depth (in mm) inside the combustion chamber at varying spray angle.

Historical Background: Emission control

The very first indication of exhaust problem was observed during early 1950s in Los-Angeles region of USA, where transport vehicles were responsible for the formation of photochemical smog due to the emission of Unburned Hydrocarbons and Nitrogen Oxides. The initiative against emissions from vehicles and achieved milestones are listed in following:-

Table 1

Year	Achievements and Events
1952	The photochemical reactions between Unburned Hydrocarbon and Nitrogen Oxides were explained by Prof. A.J Hagen from University of California. He demonstrated that, these reactions are responsible for the formation of photochemical smog observed in Los-Angeles.
1965	The first vehicle emission regulation and laws made in California, USA.
1968	The emission regulation was set for the all states of USA.
1970	European countries prepared vehicle emission standards.
1974	The catalytic converter was discovered for the oxidation of Unburned Hydrocarbon and Carbon monoxides in USA to achieve emission targets.

Revised Manuscript Received on February 05, 2020.
* Correspondence Author
Rahul Kumar Singh*, PhD Scholar, Mechanical Engineering Department, Rabindranath Tagore University Bhopal-464993, M.P., India, E mail id: rahulkumarsingh77@gmail.com
Dr. Dinesh Kumar Soni, Associate Professor, Mechanical Engineering Department, Rabindranath Tagore University Bhopal-464993, M.P., India, E mail id: dinsoni2419@gmail.com

1981	The three way catalytic converter was invented for simultaneous reduction of CO, HC and NO_x emission from vehicles.
1992	The catalytic converter was implemented on gasoline vehicles to achieve EURO-1 norms in European countries.
1994	The catalytic converter was improved for the vehicles, working under lean mixture operating condition.
1994	The emission norms in USA were redefined to achieve reduction CO, HC and NO_x emission by 96%, 97.5% and 90% respectively in US Tier-1 standard.
2000-2005	Numerous researches were carried out to achieve target of US Tier-2 by reducing CO, HC and NO_x emission up to the level of 98%, 99% and 95% respectively.

Historical background of emission control (N-69)

Significant progress has been made during the last decade in the area of emission control.

Many researchers claim that the most of the part of entire pollutions is in the form of emissions from compression ignition engines. They justified their claims due to the use of major propulsion power source for land and marine transport applications. The high rate of pollutions affects human life and the environment adversely. As per their research the control of emissions seems to be the possible way to reduce pollutions from environment. Based on history of emission control, some methods have potential to reduce emissions from diesel engine, are discussed as follows:-

a) The enhanced air fuel mixing due to change in design of piston geometry leads to improved combustion process and reduce emission parameters.
b) The air fuel mixing is also affected by swirl method to reduce emissions from diesel engine.
c) The emissions can also be reduced by using exhaust gas recirculation method in compression ignition engines.
d) The most popular method in the present scenario is the use of bio fuel with diesel to reduce emission from CI engines. Simultaneously, alcohols and water emulsified diesel have received attention in this regard too.
e) The optimization in spray parameter is helpful for better combustion process because of enhanced air fuel mixing.

Problem statement

Combustion of diesel fuel is used to produce high power in transportation sector. High emission from diesel engine is a major concern in terms of health and environmental aspects. The present work is aimed to conduct an investigation on combustion, emission and performance characteristics by varying nozzle depth at different spray angles through numerical simulation approach on a diesel engine.

The motivation of present research is gained by keeping the mentioned aim in mind. The present paper illustrates the effect of varying spray angle at different nozzle depth of a single cylinder diesel engine. This method is applied in present research to reduced emission from diesel engine and for improves performance of diesel engine. It will be an interesting discussion to analyze the influence of varying spray angle at different nozzle depth on all emission parameters, combustion parameters and performance parameter.

The present investigation will involve the optimization of spray angle and nozzle depth. The results from the optimization process will be analyzed to justify the object of the present research.

Research Methodology

Commercially available numerical simulation software knows as AVLFIRE is used for this research. Present research work consist a numerical simulations approach for the optimization of spray angle. The spray angles are categorized in three cases to achieve an optimum case, furthermore, four nozzle depth parameters are also categorizes under each case of spray angle. At the end of analysis the combinations of spray angle and nozzle depth are examined for satisfactory (compromised) outcomes of research.

The research methodology can be presented as following flow diagram:

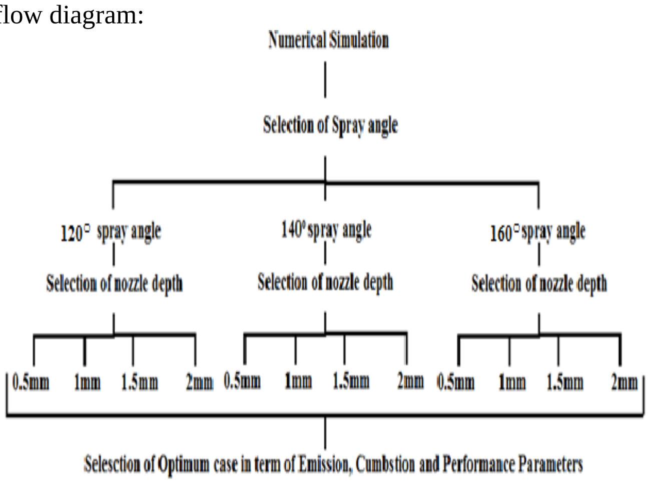


Figure 1.1 Schematic flow diagram of research methodology

II. REVIEW OF LITERATURE

The varying nozzle depth at different spray angle engine is used extensively in the field of IC engine research. Many researchers are focused on that engine due to its easiness of operation. Some promising researches are present in the literature review.

Soni, et al. [1] analysed performance combustion and emission parameters by comparing hemi- spherical bowl piston geometry and re-entrant piston bowl geometry. It was concluded that, re- entrant piston geometry is better than hemi- spherical bowl piston geometry for the decrease of NO emissions. Other parameters were also compared, but the re- entrant piston geometry was again batter then hemispherical piston geometry. Furthermore it can be used for soot mass fractions with 160° spray angle, but minimal conciliation has to be well-thought-out for performance strictures.

Soni, et al. [2] investigated the effects of three emission reduction methods numerically on a single cylinder diesel engine.

The three methods were namely,
change in swirl ratio, amount of recirculation of exhaust gases in Exhaust Gas Recirculation (EGR) procedure and addition of methanol in several quantities to prepare diesel- methanol blends. Detailed simulation analysis proves that methanol blended diesel fuel is much promising to reduce emissions from the engine.

Soni, et al. [3] analysed emission parameters by using water nano emulsion method in methanol-blended diesel fuel.

The diesel-methanol blend was categorized in three different blends to find an optimum blend. The optimum blend of diesel methanol blend was treated with water nano emulsion blend. It was concluded that, the water nano emulsion method has ability to decrease NO_x emission further from the optimum blend of diesel-methanol.

J. Li, et al. [4] investigation done on three dissimilar bowl geometries specifically: Hemispherical Combustion Chamber (HCC), Shallow depth Combustion Chamber (SCC), and the baseline Omega Combustion Chamber (OCC) were shaped with the same compression ratio of 18.5. To simulate the combustion method, computational fluid dynamics (CFD) modeling based on KIVA-4 code was accomplished. The assessments amongst three bowl geometries were accompanied in terms of velocity vector field, cylinder pressure, HRR, AHRR, temperature delivery in the combustion chamber and the mass fraction of emissions (CO and NO) at engine speeds of 1200, 2400 and 3600 rpm.

Prasad, et al. [5] thorough three-dimensional CFD simulations connecting flow and combustion chemistry are used to learning the consequence of swirl tempted by re-entrant piston bowl geometries on contaminant emissions from a single-cylinder diesel engine. The standard engine outline contains of a hemispherical piston bowl and an injector with finite sac volume. The combustion chamber geometries were created for in cylinder air motion and an optimum geometry selected in terms of swirl motion and turbulence kinetic energy (TKE) for in compression top dead centre (TDC). The optimal nature of this re-entrant piston bowl geometry is established by thorough combustion simulations and emission expectations.

Rakopoulos, et al. [6] in the current work, a firstly effort is made to associate two recent engine simulation models established by the authors. The firstly one is a CFD model advanced from scratch and the second one is a thorough quasi-dimensional model. These two models have been apply to simulate the closed part of the cycle of a HSDI diesel engine functioning under motoring situations, expending three piston bowl geometries for three engine rotational speeds of 1500, 2000 and 2500 rpm. These substitute geometries are created by changing the ratio of piston bowl diameter to cylinder diameter (d/D) from 64% (which is the standard case) to 54% and 44%, accumulative respectively the piston bowl height so as to retain the compression ratio constant.

Ke Li, et al. [7] in this paper, a model of the free piston engine successively similar charge compression ignition combustion below numerous piston trajectories is accessible. The idea of trajectory-based combustion control is projected. Simulation outcomes signpost undoubtedly that with the capability of accurate piston trajectory tracking, the FPE is capable to adjust the complete combustion procedure by varying the volume outline of the combustion chamber and consequently changing the in-cylinder gas temperature and pressure traces, the indicated output work and the heat loss.

Yao, et al. [8] the initially thing paid consideration to is that a great deal of essential hypothetical investigation has been carried out. Firstly, numerical simulation has turn out to be a moral remark and a powerful tool to examine HCCI and to progress control approaches for HCCI since of its better flexibility and lower cost associated with engine investigates. Five types of models functional to HCCI engine modeling are deliberated in the current paper. Second, HCCI can be applied to a variation of fuel varieties. Combustion phasing and procedure range can be controlled by the alteration of fuel features. Third, it has been comprehended that progressive control approaches of fuel/air mixture are more significant than simple similar charge in the procedure of the monitoring of HCCI combustion procedures.

Bari, et al. [9] this investigation began with enhancing the design of the guide vanes over the simulation technique. Nevertheless, the optimization was inadequate only to the quantity of guide vanes while vane height, angle, and length were retained endless. Afterward investigating the simulation consequences of in-cylinder airflow features of TKE (turbulence kinetic energy), velocity, vortices, and swirling strength of 10 guide vanes models of dissimilar vane quantities that varied in between 3 and 12, the model with 4 vanes was initiate to be the finest one. Afterward that, five guide vanes models with the quantity of vanes ranging amongst 3 and 7 were invented and verified one by one on a CI engine run with biodiesel along with a normal CI engine run with biodiesel and petro-diesel having no vanes.

S. Scott, et al [10] A unrestricted piston, internal combustion (IC) engine, functioning at high compression ratio (~30:1) and low equivalence ratio (f~0.35), and applying similar charge compression ignition combustion, has been projected by Sandia National Laboratories as a means of suggestively Refining the IC engine's cycle thermal efficiency and dissipate emissions. The engine was examined through a zero dimensional thermodynamic model approach, where friction elements heat transfer and experimental scavenging models has been used for the functioning of engine. The cycle simulations expending hydrogen as the fuel, have designated the precarious factors upsetting the engine's enactment, and recommend the limits of enhancement promising comparative to conventional IC engine technologies.

After a detailed discussion, it can summarize that performance and emission analysis has not carried out by any researcher in varying nozzle depth at different spray angle in view of emission parameters, combustion parameter and performance parameters. Now, it is very much essential to investigate the effects of variation in nozzle depth and spray angle due to its importance in terms of performance and emission characteristic.

Research Objectives

The performance and emission characteristics will be analyzed numerically for a single cylinder kirloskar diesel engine by varying nozzle depth at different spray angle the objectives of this work are as follows.

1. The nozzle depth will be changed from 0.5mm to 2mm and spray angle will be change from 120^o to 160^o.
2. The optimum nozzle depth and spray angle will be founded by analyzing performance and emission characteristics of diesel engine. Commercially available simulation software will be used for numerical simulation.
3. The present research illustrates the effect of variation in nozzle depth at different spray angle on a kirloskar single cylinder diesel engine.

The researchers all around the world are trying to reduce heat loss from the CI engine, thus it requires more attention to analyze the heat balance at varying nozzle depth at different spray angle engines and to check there effects on thermal efficiency and power at various nozzle depth at different spray angle of nozzle. There is need to explore the optimum nozzle depth and optimum spray angle on which engine gives best result.

III. SURVEY OF NUMERICAL MODELS

Description of single cylinder diesel engine

A commercially available single cylinder diesel engine (model TV-1) was used for simulation purpose. This engine has a hemispherical piston bawl geometry with a standard spray angle of 120^0 and nozzle depth 1mm. the parameters regarding engine geometry and working conditions are mentions in reference [1]. Some of the important design parameters are listed below in the table 3.1

Table 3.1 Specification of simulated diesel engine

Make	Kirloskar engine, TV-1
Number of cylinders	1
Bore X Stroke	87.5 mm x110 mm
Swept volume	661 cc
Clearance volume	40.1cc
No. of nozzle holes	3
Connecting rod length	234 mm
Rated output	5.2 kw
Cylinder head curvature	Flat
Peak pressure	72-76 bar
Clearance	0.112 cm = 1.2 mm
Type of combustion chamber (Piston bowl shape)	Hemispherical open combustion chamber
Rated speed	1500 rpm
Compression ratio	17.5: 1 (Standard engine) & 15:1 – 18:1(VCR)

Assessment of Numerical Models

The Computational fluid dynamics (CFD) is well-defined as a division of fluid mechanics which contains numerical approaches and algorithms. The CFD along with arithmetical approaches and algorithms use to resolve and examine difficulties that comprise fluid flows. The CFD codes are used concluded computers and it covers millions of equations to perform simulation of contact amongst liquids and gases with surfaces well-defined by boundary situations. The estimated solutions can be attained in furthermost of the cases uniform with the use of high-speed super computers. Computational fluid dynamics permits the learning of flow dynamics in side thermal systems. By applying CFD, one can produce its own codes to signify preferred thermal system. The software will deliver forecasts of fluid dynamics by applying use of fluid physics and chemistry to the essential prototype.

Consequently, CFD is a refined computationally-based design and examination method. CFD software can simulate affecting bodies, multiphase physics, chemical response, flows of gases and liquids, heat and mass transfer, fluid-structure interface etc. A 'virtual prototype' of anticipated system can be built by applying CFD software to examine and it will deliver performance of the design in positions of image and data. The continuing investigation is attentive on to progress the correctness and speed of complex simulation situations such as turbulent flows. The simple physical laws of preservation of energy, momentum and mass are used to produce governing equation of flow dynamics by coupling non-linear partial differential equations. Firm expectations are made approximately the belongings of the fluid intricate to attain analytical resolutions of simple flow dominions. For conservative design of equipment, devices, and structures used for controlling fluid flow patterns, designers have to depend on empirical formulae, rules of thumb, and experimentation. The simple stages of CFD analysis can be classified in subsequent three sections.

a) Preprocessing

The initially tactic to resolve all CFD difficulties is identified as Preprocessing. It covenants with the category of flow involved in the simulation, formation of geometry and functioning situations and application of renowned physical phenomena.

b) Solver

The second stage to resolve CFD difficult is identified as solver. By setting solver section in CFD, the set of algebraic equations are shaped from fractional differential equations (PDE). It includes mathematical equations of flow model, arrangement of forces functional on a body, computational domain and conservation equations for the mass momentum and energy. The discretization method is similarly a part of solver. The dissemination of complete geometry in to cells (mesh generation) by applying triangular, quadrilateral and hexahedron mesh with structured, unstructured and hybrid grids as shown in **Fig. 3.1.**

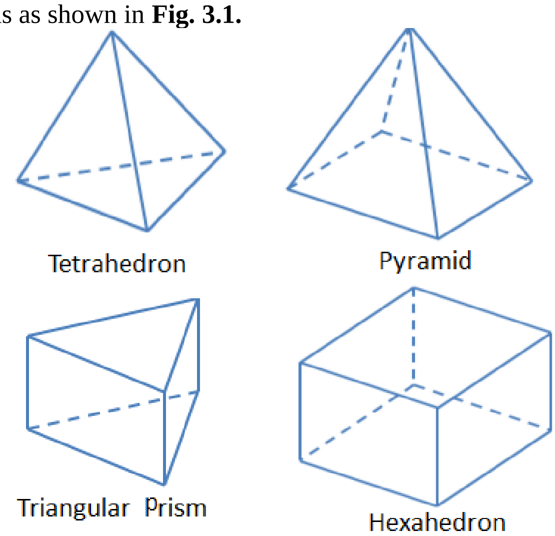


Figure 3.1 Types of mesh

IV. METHODOLOGY

In the current examination, numerical simulation is performed by using a simulation code AVL FIRE for a single cylinder, DI diesel engine having a hemispherical bowl piston. This software uses finite volume discretization scheme to resolve the simulation procedure.

Additionally, it has four modules to resolve dissimilar kinds of engine simulations, like AVL Boost, AVL Cruise, AVL Excite and AVL FIRE. The current examination uses AVL FIRE simulation code to solve the combustion simulation. This simulation code runs software for two strokes of the diesel engine i.e. compression stroke and expansion stroke. Consequently, simulation can run for the compression stroke starting at 540˚crank angle to expansion stroke ends at 900˚ crank angle; whereas, 720˚crank angle is considered as a TDC (top dead center). The AVL FIRE simulation software is shown in **Fig. 4.1** and schematic diagram of numerical simulation is shown in **Fig. 4.2.**

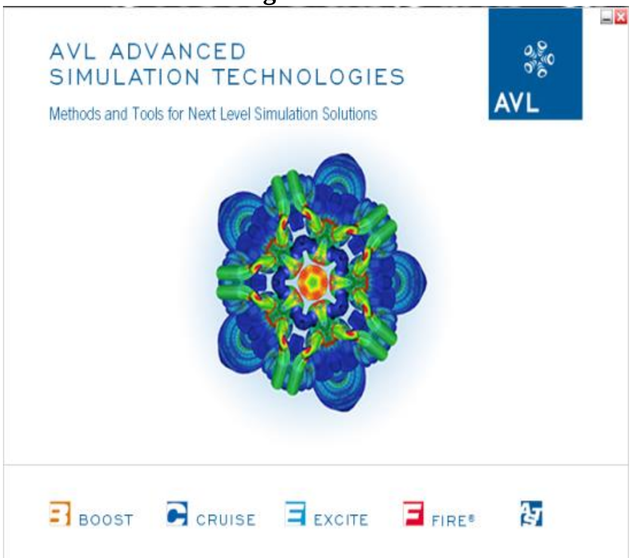


Figure 4.1 AVL FIRE simulation software

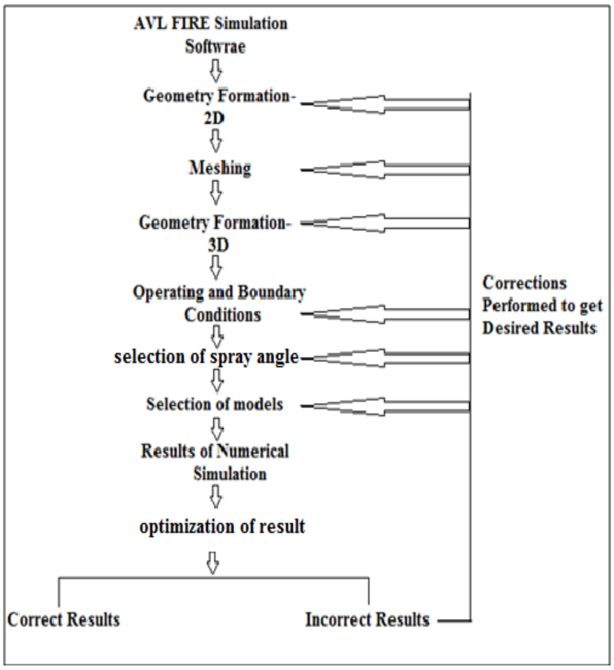


Figure 4.2 Schematic diagram of Flow process of Numerical simulation

Some elementary structures of AVL FIRE are listed below.

a) Steady and transient simulation mode.
b) Compressible and incompressible, laminar and turbulent flows.
c) Handling of domains with and without moving boundaries.
d) Mixing plane and sliding interfaces for rotating parts.
e) User-defined formulas and functions.
f) Automated mesh generation through ESE diesel.

The ESE diesel is a module of AVL FIRE, accessible in the software, which is used to simulate diesel engine. The modeled engine sample is a 1.9 L, single cylinder, DI diesel engine. The basic input factors of à diesel engine like engine type, number of cylinders, bore, compression ratio, crank radius, connecting rod length and stroke are providing at the preliminary stage of the simulation procedure. These standards can be inserted at the appropriate place of the ESE diesel module

The value of preliminary pressure and initial temperature along with other factors is selected for simulation from reference [1], on the basis of experimental outcomes as mentioned in **Table 4.1**. Furthermore, for the whole set of simulation, compression ratio, bowl volume, mass injected and speed of the engine remains constant. The values of initial and boundary condition can be implanted in the preferred location of the software as shown in **Fig. 4.9.**

Table 4.1 Initial / Boundary conditions

Initial pressure	0.65 mpa
Initial temperature	300 k
Piston temperature	550 k
Liner temperature	425 k
Head temperature	475 k
Fuel injection timing	23° CA BTDC
Fuel spray angle	120°
Injection type	Single injection
Mass of fuel injected	1.6e-05 kg/cycle
Nozzle depth inside combustion chamber	1mm

Grid independency test

In numerical simulation of any problem, it is actual significant to ascertain that the models used are valid for the actual physical circumstance which are simulating and computational domain used for calculation is independent of mesh size and number of cells. To confirm whether the mesh chosen is independent of grid size, a grid independency test is conducted; three different meshes are treated for test with 34287, 45667 and 56109 cells. The predictions are well within acceptable limits and simulation time is approximately 6 h.

Hexa hedral mesh of hemi spherical bowl geometry with 40868 cells is created by using AVL FIRE ESE diesel module. Three differentcell sizes 34287, 45667 and 56109 of same geometry compared and result shows good agreement as shown in **Fig. 4.3**. So, the simulation was conducted with geometry of 45667cells to reduce process timing. Thus, the simulation is carried with the mesh of 45667 cells for further simulation.

Table 4.2 Grid independency test

No. of cells at TDC in the domain	Maximum pressure (bar)	% deviation
34287	56.47	-
45667	54.74	3.10
56109	54.26	0.87

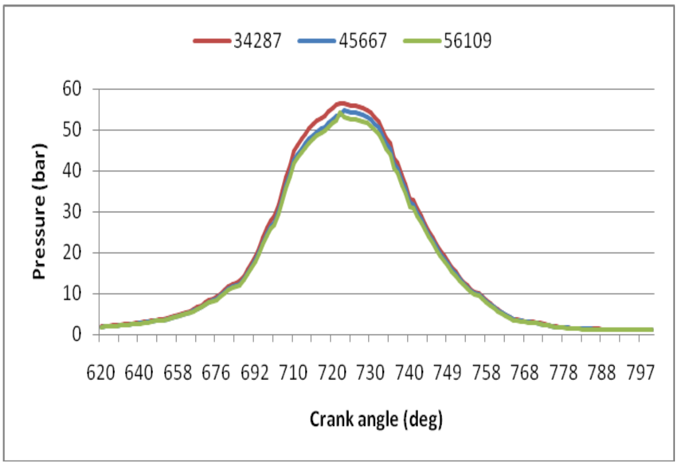


Figure 4.3 Comparison of pressure results of all cell size

V. RESULT & DISCUSSION

The results of numerical simulation analysis are discussed for single cylinder diesel engine which operated under the varying spray angle at different nozzle depth. The standard operating conditions of the used engine are 120° spray angle and 1mm nozzle depth. Thereafter the nozzle depth will Change from 0.5mm to 2mm (such as 0.5mm, 1mm, 1.5mm and 2mm) and spray angle between 120° to 160° (such as 120, 140 and 160°). The effects of numerical simulation have analyzed in terms emission parameters, and presented graphically for better understanding in the following subsections. On the basis of analysis, an optimum nozzle depth and spray angle will be selected.

Selection of optimum result from each case

In this section, the results of different emission parameters, are combined and discussed to select the optimum result. The optimum result will be selected on the basis of minimum emission produce during the process. The optimum result of emissions will be considered from each set of CO, NO, Soot and HC emissions from different spray angle (120°, 140°, 160°) and varying nozzle depth (0.5mm, 1mm, 1.5mm, 2mm).

Comparison of result for CO emission from each case

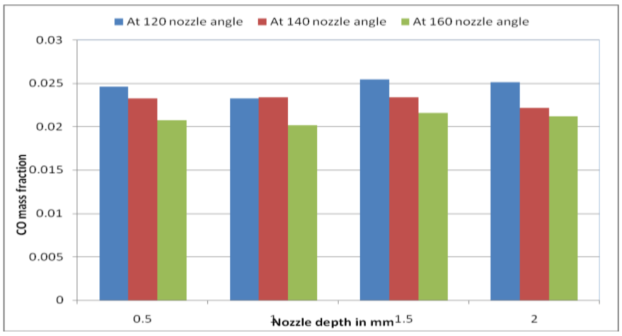
In the **fig. 5.1**, by comparing CO emission at different nozzle depth and different spray angle, it can be analyzed that , the amount of CO emission is minimum at 1mm nozzle depth and 160° Spray angle. The value of CO emission is 0.020161 mass fractions, which is lower than the other values of CO emissions. This value of CO emission at different spray angle and at different nozzle depth can be seen in **Table-5.1.**

Table-5.1 CO emission at different spray angle and at different nozzle depth

Nozzle depth	At 120 nozzle angle	At 140 nozzle angle	At 160 nozzle angle
0.5	0.024564	0.023248	0.020739
1	0.023248	0.023346	0.020161
1.5	0.025425	0.023393	0.021597
2	0.025116	0.02214	0.021165

Figure 5.1Comparison of result for CO emission

Comparison of result for NO emission from each case



In the **fig. 5.2**, by comparing NO emission at different nozzle depth and different spray angle, it can be analyzed that, the amount of NO emission is minimum at 2 mm nozzle depth and 120° Spray angle. The value of NO emission is 0.000313 mass fractions, which is lower than the other values of NO emissions. This value of NO emission at different spray angle and at different nozzle depth can be seen in **Table-5.2.**

Table-5.2 NO emission at different spray angle and at different nozzle depth

Nozzle depth	At 120 nozzle angle	At 140 nozzle angle	At 160 nozzle angle
0.5	0.000327	0.000447	0.000355
1	0.000447	0.000449	0.000414
1.5	0.000343	0.000408	0.000369
2	0.000313	0.000396	0.000319

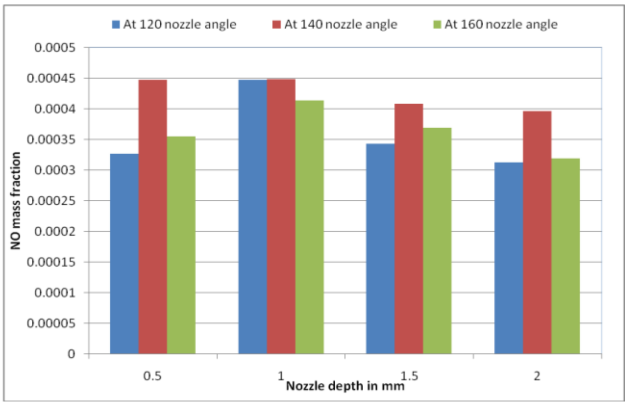


Figure 5.2 Comparison of result for NO emission

Comparison of result for Soot emission from each case

In the **fig. 5.3**, by comparing Soot emission at different nozzle depth and different spray angle, it can be analyzed that, the amount of Soot emission is minimum at 2 mm nozzle depth and 140° Spray angle. The value of Soot emission is 0.0000350 mass fractions, which is lower than the other values of Soot emissions.

This value of Soot emission at different spray angle and at different nozzle depth can be seen in **Table-5.3.**

Table-5.3 Soot emission at different spray angle and at different nozzle depth

Nozzle depth	At 120 nozzle angle	At 140 nozzle angle	At 160 nozzle angle
0.5	3.90E-05	3.87E-05	6.14E-05
1	3.87E-05	3.5253E-05	0.00005587
1.5	3.99E-05	3.86E-05	6.10E-05
2	3.85E-05	3.50E-05	5.5471E-05

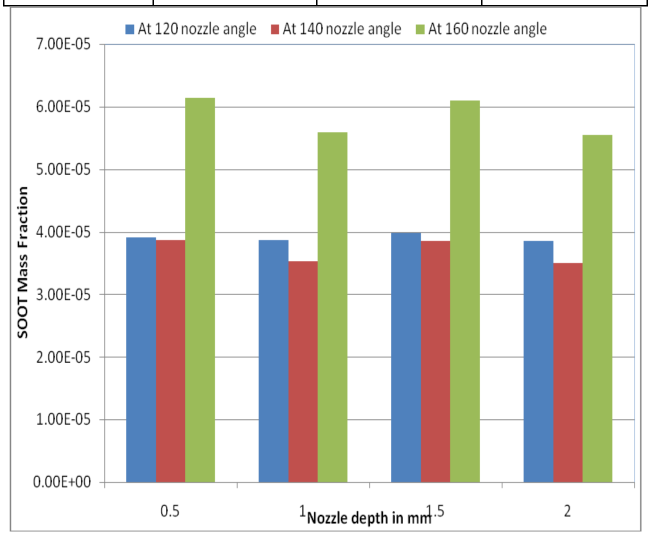


Figure 5.3Comparison of result for Soot emission

Comparison of result for HC emission from each case

In the **fig. 5.4**, by comparing HC emission at different nozzle depth and different spray angle, it can be analyzed that, the amount of HC emission is minimum at 1 mm nozzle depth and 160° Spray angle. The value of HC emission is 0.016035 mass fractions, which is lower than the other values of HC emissions. This value of HC emission at different spray angle and at different nozzle depth can be seen in **Table- 5.4.**

Table-5.4

Nozzle depth	At 120 nozzle angle	At 140 nozzle angle	At 160 nozzle angle
0.5	0.016306	0.016202	0.016053
1	0.016202	0.016275	0.016035
1.5	0.016653	0.016332	0.016831
2	0.016544	0.016924	0.016661

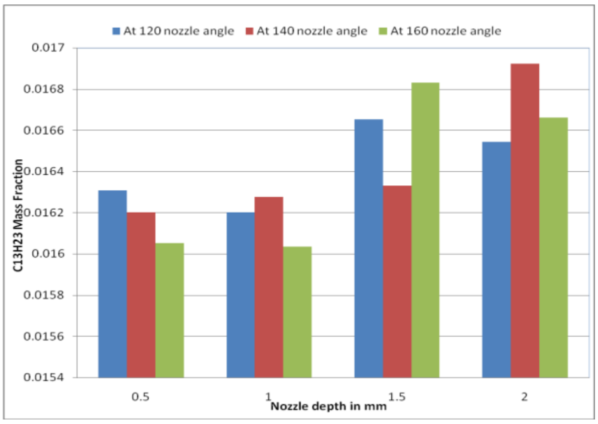


Figure 5.4 HC emission at different spray angle and at different nozzle depth Comparison of result for HC emission

VI CONCLUSION & FUTURE SCOPE

The following conclusions are obtained from above results which are:-

- The minimum mass fraction of CO emission is observed at 160° spray angle with 1mm nozzle depth the value of minimum CO emission is 0.020161 mass fractions. The maximum value of CO emission is observed at 120° spray angle with 1.5mm nozzle depth. the value of maximum CO emission is 0.025425 mass fraction. The percentage reduction in CO emission is from maximum value to minimum value is 20%.
- The minimum mass fraction of NO emission is observed at 120° spray angle with 2 mm nozzle depth the value of minimum NO emission is 0.000313 mass fractions. The maximum value of NO emission is observed at 140° spray angle with 1 mm nozzle depth. the value of maximum NO emission is 0.000449 mass fraction. The percentage reduction in NO emission is from maximum value to minimum value is 30%.
- The minimum mass fraction of Soot emission is observed at 140° spray angle with 2 mm nozzle depth the value of minimum Soot emission is 0.0000350 mass fractions. The maximum value of Soot emission is observed at 160° spray angle with 0.5 mm nozzle depth. The value of maximum Soot emission is 0.0000614 mass fraction. The percentage reduction in Soot emission is from maximum value to minimum value is 42%.
- The minimum mass fraction of HC emission is observed at 160° spray angle with 1 mm nozzle depth the value of minimum HC emission is 0.016035 mass fractions. The maximum value of HC emission is observed at 140° spray angle with 2 mm nozzle depth. The value of maximum HC emission is 0.016924 mass fractions. The percentage reduction in HC emission is from maximum value to minimum value is 5%.

The results showed that the emission parameter and depends on the spray angle and nozzle depth. To get suitable emission result from the above case some compromise has to be considered for performance and combustion parameters.

Future Scope

Researchers and engineers around the world are focusing more on emissions from diesel engines owing to serious environmental issue and understandably ready to compromise with power and efficiency. The present research showing an simulation work to achieve low emissions. The present research can be further enhanced by using combination of other spray parameters with spray angle and nozzle depth. The presented work is numerical simulation approach which can be further treated in an experimental work by considering it's cost analysis.

REFERENCES

1. Dinesh Kumar Soni, Rajesh Gupta. Numerical analysis of flow dynamics for two piston bowl designs at different spray angles, Journal of Cleaner Production 149 (2017) 723-734.
2. Dinesh Kumar Soni, Rajesh Gupta. Numerical investigation of emission reduction techniques applied on methanol blended diesel engine. Alexandria Engineering Journal (2016) 55, 1867–1879
3. Dinesh Kumar Soni, Rajesh Gupta. Application of nano emulsion method in a methanol powered diesel engine, Energy 126 (2017) 638e648.
4. J. Li, W.M. Yang , H. An, A. Maghbouli, S.K. Chou. Effects of piston bowl geometry on combustion and emission characteristics of biodiesel fueled diesel engines. Fuel 120 (2014) 66–73.
5. B.V.V.S.U. Prasad, C.S. Sharma, T.N.C. Anand, R.V. Ravikrishna. High swirl-inducing piston bowls in small diesel engines for emission reduction. Applied Energy 88 (2011) 2355–2367.
6. C.D. Rakopoulos *, G.M. Kosmadakis, E.G. Pariotis. Investigation of piston bowl geometry and speed effects in a motored HSDI diesel engine using a CFD against a quasi-dimensional model. Energy Conversion and Management 51 (2010) 470–484.
7. Chen Zhang, Ke Li, Zongxuan Sun. Modeling of piston trajectory-based HCCI combustion enabled by a free piston engine. Applied Energy 139 (2015) 313–326.
8. Mingfa Yao*, Zhaolei Zheng, Haifeng Liu. Progress and recent trends in homogeneous charge compression ignition (HCCI) engines. Progress in Energy and Combustion Science 35 (2009) 398–437.
9. S. Bari ,IdrisSaad. Optimization of vane numbers through simulation and experiment, and investigation of the effect on the performance and emissions of a CI (compression ignition) engine run with biodiesel. Energy 79 (2015) 248e263.
10. S. Scott Goldsborough and Peter Van Blarigan. A Numerical Study of a Free Piston IC Engine Operating on Homogeneous Charge Compression Ignition Combustion. SAE TECHNICAL PAPER SERIES 1999-01-0619.
11. StefaniaFalfaria,, Federico Brusiania, PieroPellonia. 3D CFD analysis of the influence of some geometrical engine parameters on small PFI engine performances – the effects on the tumble motion and the mean turbulent intensity distribution. Energy Procedia 45 (2014) 701 – 710 68th Conference of the Italian Thermal Machines Engineering Association, ATI2013.
12. S. Jafarmadarand M. Khanbabazadeh. A Computational Study of the Effects of Combustion Chamber Geometrieson Combustion Process and Emission in a DI Diesel Engine. Journal of Fuel and Combustion Vol. 1, No. 1, Sept. 2008.
13. YasinVarol ,Hakan F. Oztop , MujdatFirat, Ahmet Koca. CFD modeling of heat transfer and fluid flow inside a pent-roof type combustion chamber using dynamic model. International Communications in Heat and Mass Transfer 37 (2010) 1366–1375.
14. RaoufMobasheri, Zhijun Peng. CFD Investigation of the Effects of Re-Entrant Combustion Chamber Geometry in a HSDI Diesel Engine. International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering Vol:7, No:4, 2013.
15. Semin, Rosli Abu Bakarand Abdul Rahim Ismail. Computational Visualization and Simulation of Diesel EnginesValve Lift Performance Using CFD. American Journal of Applied Sciences 5 (5): 532-539, 2008ISSN 1546-9239.
16. Abdul Gafoor C.P. , Rajesh Gupta. Numerical investigation of piston bowl geometry and swirl ratio on emission from diesel engines. Energy Conversion and Management 101 (2015) 541–551.
17. Horng-Wen Wu ,Shiang-WuuPerng. Numerical analysis of thermal turbulent flow in the bowl-in-piston combustion chamber of a motored engine. International Journal of Thermal Sciences 43 (2004) 1011–1023.
18. B. Yadollahi, M. Boroomand. The effect of combustion chamber geometry on injection and mixture preparation in a CNG direct injection SI engine. Fuel 107 (2013) 52–62.
19. Shengli Wei, KunpengJi, XianyinLeng, Feihu Wang, Xin Liu. Numerical simulation on effects of spray angle in a swirl chamber combustion system of DI (direct injection) diesel engines, Energy 75 (2014) 289e294.
20. ZvonimirPetranovi_c*, Milan Vujanovi_c, Neven Dui. Towards a more sustainable transport sector by numericallysimulating fuel spray and pollutant formation in diesel engines. Journal of Cleaner Production 88 (2015) 272e279.
21. Sibendu Soma, Anita I. Ramirez , Douglas E. Longman , Suresh K. Aggarwal. Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel engine conditions. Fuel 90 (2011) 1267–1276.
22. M. Gorji-Bandpy, S. Soleimani, and D. D. Ganji. The Effect of Different Injection Strategies and Intake Conditions on the Emissions Characteristics in a Diesel Engine.

AUTHORS PROFILE

Rahul Kumar Singh is pursuing P.hD studies in Mechanical Engineering branch Rabindranath Tagore University, Bhopal Madhya Pradesh India. As a part of thesis submission this work on Internal Combustion Engine conducted under the guidance of Dr. Dinesh Kumar Soni.

E mail id: rahulkumarsingh77@gmail.com

Dr. Dinesh Kumar Soni Currently he is working as an Associate Professor, Mechanical Engineering Department, Rabindranath Tagore University Bhopal-464993, M.P., India, He wrote good number of Technical papers in the areas of Internal Combustion Engine. He has good years of professional experience in Teaching. E mail id: dinsoni2419@gmail.com