SEMARAK ILMU PUBLISHING

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Review Analysis on CFD Techniques and Numerical Methods for IC Engines Fueled with Diesel and Biofuels

Bavesh Pathak[1,*], Asfakahemad Shekh[1], Nikul Patel[1], Vimal Patel[1]

[1] Department of Mechanical Engineering, Faculty of Technology & Engineering, Maharaja Sayajirao University of Baroda, Gujarat 390001, India

ARTICLE INFO

Article history:
Received 19 June 2023
Received in revised form 8 September 2023
Accepted 16 September 2023
Available online 4 October 2023

Keywords:
CFD; Turbulence; IC Engine; Biodiesel; Combustion Chamber

ABSTRACT

Over the years, there has been a drastic reduction in petroleum-based fuel stocks and stringent emission requirements for internal combustion engines by environmental authorities around the world. A marginal improvement in efficiency and emissions shows significant benefits. Fluid motion within the combustion chamber plays a vital role and is considered a crucial parameter for controlling the combustion process. The fuel supply system or induction system of the Spark Ignited engine prepares and controls air-fuel mixing. The system in the SI engine is capable of generating bulk motion and turbulence prior to the ignition process. In the CI engine Induction system (Swirl and Tumble), the shape of the chamber (Squish) controls the path and quality of air whereas the diesel injection system controls Spray characteristics (Penetration, Atomization, Break-up, and Evaporation of Spray). During the last three decades, a lot of efforts have been made by researchers to develop multidimensional fluid dynamics codes for the prediction of flow in an engine for reducing engine development time and cost. To be a more trusted and acceptable tool, the results of numerical analysis of the combustion chamber. and its fluid dynamics have to show accuracy in the prediction of emission standards and combustion parameters by taking the minimum possible time. This article reviews the methods and results of numerical analysis and CFD results of CI engines run on diesel and biofuels. It also reviews results and their accuracy for different CFD.

1. Introduction

Atmospheric air which is polluted due to pollutants from internal combustion engines, majorly in highly populated areas, is the main criterion of concern for protecting the environment [1]. Designed specific internal combustion engines should be developed in such a manner to meet emission standards globally by minimizing poisonous concentrations in smoke and by minimizing fuel consumed [2]. This task can be achieved by controlling the combustion process, adding additives, and normalizing exhaust gas emissions in the afterburning process [3-5]. For a long, it has been observed that fluid motion or turbulence within the combustion chamber is the most important parameter which controls the combustion process. Unfortunately, it is still not well understood to how much extent this parameter influences the combustion phenomenon. Preparing a mixture of air-fuel and

[*]*Corresponding author.*
E-mail address: gecbhbcp@gmail.com

https://doi.org/10.37934/arfmts.110.1.4062

controlling flammability through mixing air-fuel at an optimized level needs knowledge of fluid mechanics. Because of large density variation and many other parameters like viscosity, conductivity, and also other parameters compared to diesel as a reference, it seems knowing fluid flow parameters is one of the challenging tasks nowadays. In SI engine fuel supply system or induction system prepares and controls air-fuel mixing. The system in SI engine is capable of generating bulk motion and turbulence prior to the ignition process [2,6]. In CI engine induction system, the shape of the piston bowl controls air motion whereas the mechanism for the fuel injection system decides spray dynamics, which includes fuel penetration, fuel atomization, fuel break-up, and fuel evaporation. Knowing the behavior of the flow pattern is important for knowing the heat transfer rate across the cylinder wall, turbulent mass transport, and potential of momentum. To seek improvement in the design of an IC engine, one needs to understand the characteristics of the engine in terms of turbulence phenomena, its interaction, and bulk flow. The flowing nature within a cylinder is very complex due to its turbulent, reactive, unsteady, and three-dimensional behavior.

The objective of this review is to identify available codes for both commercial and non-commercial turbulence models which can benefit researchers in the combustion field. It is also aimed at listing the pros and cons of different models and identifying the extent of simplicity and complexity which can form a basis for research in the field of combustion. Researchers in the combustion-specific field in the present scenario find difficulties choosing correct codes or turbulence models as there is no single model that can be readily applied to all turbulent flows without a degree of adjustment and experience in selecting the most appropriate model, which in turn motivates them to compare codes and models on various parameters. This can be helpful for researchers to select optimized codes and models for combustion studies. In addition, tuning the model coefficients is of the utmost importance to obtaining accurate, consistent, and robust simulations.

2. In Cylinder Flow Techniques

Power output from IC engine can be optimized in many ways. One of them is to work on techniques used for the in-cylinder flow. Flow parameters of any combustion chamber used with varying capacities and also various applications have complex physical and chemical characteristics [7]. It consists of 3D dynamics of evaporating fuel sprays in transit conditions. This fuel flow collides with various components of gases and undergoes igniting, chemically reacting, mixing, and transferring heat thereof. Numerical methods are capable of calculating such complicated fluid dynamics within a combustion chamber with arbitrarily designed piston head geometry, turbulence, and cylinder wall heat transfer. Different methodologies have been adopted to achieve the desired results by the technique of inserting fresh air at an optimized angle into the combustion process to pursue turbulence characteristics like tumble and swirl [8-10]. Various numerical methods and CFD codes are capable of solving unsteady state conditions of turbulent flow and chemical reaction analysis for a mixture of an ideal gas. This article is prepared to sum-up analytical flow measurement in view of numerical and CFD simulation studies. Various turbulent models are adopted to forecast accurate flow field parameters by taking differently configured engines and by taking governing equations on those turbulent models.

2.1 Turbulence models-CFD

The level of turbulence decides the quality of combustion because more turbulence can mix more air-fuel charge and thus have a positive effect on in-cylinder flow patterns. Recently, CFD has become the most powerful technique for the study of flow and its parameters. CFD techniques can study moving particles and particle behavior which usually involves many complex calculations and many numbers of other variables [11,12]. This technique is based on a finite volume mathematical model. Here, this, first in computational domain flow is modeled by CFD modeling tools. Now the modeled area is converted into many small elements, which is known as meshing, or also can be called a discretization process. All those small elements are considered as a control volume and all these possess the same properties which are possessed by fluid as a whole. Some of those properties are mass, momentum, and energy [13]. According to the flow which is set to be studied, one can choose said properties to control volume. Basic calculations which work with CFD techniques or while using CFD software are partial differential equations. The PDE can show the physical identity of flow in terms of theoretical equations. Launder and Spalding proposed the traditional k-Ɛ model. It has gained a lot of traction since it produces reliable findings, is computationally efficient, and has effective prediction characteristics for a wide variety of turbulent flows. The k-Ɛ model consists of a standard model, a renormalized group (RNG) model, and a realizable model. Kinetic energy and its dissipation rate in turbulence are considered variables in this model. It applies two transport partial differential equations. The basic k-Ɛ model is semi-empirical and the basis of it is model transport equations [14-17]. The model is derived by assuming the flow as purely turbulent and by taking the effects of molecular viscosity at a minimal level. For free-flowing streams, it is thought to be quite accurate. The RNG k-Ɛ model is almost the same and as effective as the basic or standard k-Ɛ model, but with notable differences. RN in RNG refers to renormalization. This technique was used to create the RNG model. As a result, there are different constants, new terms and functions presented compared to the Standard k-Ɛ model. In Ansys software, one can find the usage of the k-ω model for specific tasks. The k-ω model here applies a modified Wilcox model to predict flow behaviors which include mixing layers, a variety of jets, and shear spreading rates by keeping low Reynolds number effect, compressibility factors, and other parameters. The said characteristics make use of forecasting flow properties for flow near the boundary wall and free shear flow condition. k- ω is an empirical model constructed based on realistic experimental data for model transport equations and, due to that, only it can provide accurate results for the near-wall region. In flow cases in the studies of shock waves and airfoils, SST k- ω is proven to be the best-suited model. In the SST k- ω model, first 's' refers to shear, second 's' refers to stress, and 't' refers to transport. The model was created by Menter in which the precision and effectiveness of 'k- ω' and 'k-Ɛ' models combine to predict effective flow properties with accuracy. The accuracy of this model is achieved by the use of blending factors on the scale of '0' and '1'. Here '0' implies being away from the boundary, whereas '1' implies proximity to the boundary. Cross-diffusion derivative terms are used by the SST model. The other method for predicting flow behavior with a turbulence model is large eddy simulations [18,19]. The method adopted to reduce challenges occurs from direct numerical simulations (DNS). The DNS method is comparably tedious and expensive while solving complex geometry and complex boundary conditions as it uses Navier-Stokes equations without a turbulent model [20-22]. For choosing any of the above models for research or experiment, one needs a basic understanding of flow parameters in the study and analysis of flow prediction and for that, it is primarily important for researchers to know the physics of flow, geometry of the flow field, accuracy requirement and other constraints like availability of time and software. New practices lead to combining two methodologies simultaneously, like the RANS approach. The RANS approach applies two-equation models for calculating unknown variables.

Figure 1 shows a comparison of the most widely used approaches to solving turbulent flows. The computational cost of a simulation increases from RANS to DNS concerning the increase in the number of degrees of freedom required to solve the flow. As a consequence of the computational cost, scale-resolving approaches like DNS and LES are generally applied to simple geometries, while hybrid RANS-LES, URANS, and RANS can be applied to complex industrial problems.

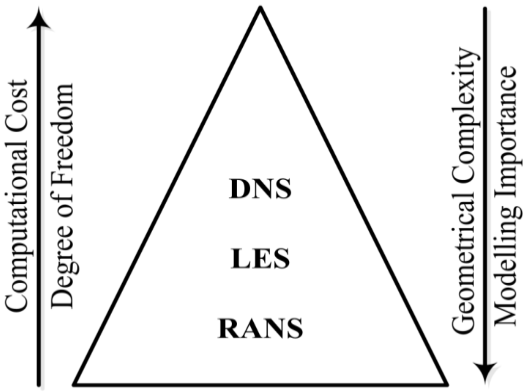

Fig. 1. Comparison of Various Approach

3. Numerical Analysis & Computational Studies on the Combustion Chamber of an IC Engine

The ability to forecast and simulate the charge and emission flow dynamics are critical for an internal combustion engine's efficiency, fuel economy, and ideal operating characteristics. Various experimental procedures have been created for this purpose during the years of engine development [23]. However, the high cost and time required, as well as the restricted potential of tests, rendered these methods outdated for engine design and development. The need for Computational Fluid Dynamics (CFD) algorithms arose from the evolution and expansion of computing capabilities. CFD codes give a numerical representation of the flow field by advancing the solution across space and time by converging solutions for the flow field after solving equations pertaining to it. Nowadays, CFD is used in practically every sector of engineering. Without the requirement for prototyping, even the most complex phenomena may be accurately reproduced by saving money and time. Different flow geometries and structures can be correctly simulated quickly along with a reasonable expenditure, leading to the popularity of CFD tool usage widely. The ability to forecast and simulate the charge and emission flow dynamics are critical for an internal combustion engine's efficiency, fuel economy, and ideal operating characteristics. Various experimental procedures have been created for this purpose during the years of engine development. However, the high cost and time required, as well as the restricted potential of test. The CFD workflow includes the following steps: a) Drafting geometry, b) generation of the grid, c) model setup, d) solving flow field, and e) results in the evaluation. The finite volume technique, based on the partial differential equations being solved in integral form, has an application in CFD codes to discrete the flow geometry. In this discretization, after the geometry is partitioned into cells, the governing equations are solved for a tiny control volume [24-26]. Experimental work asks for time and money involvement, which may be a concern for researchers. Specifically, when one needs to know about what is happening inside IC Engine and detailed combustion phenomena, in that case, there would be a lot of investment of time and money. So, numerical codes and software based on it solve the purpose effectively without much concern. Advancements in the hardware and software side of the computer CPU have brought faster speeds for the processors to process code and to converge solutions in the current era. CFD code with this advancement has proven to be the best tool for optimizing the combustion process. With at least time, nowadays it is possible to solve problems with a different variety of configurations again with only the least needed resources here. KIVA-Chemkin, Ansys, Star-CD, and GT-Power are a few codes and software available in the market to investigate the field of flow and its characteristics.

Park [27] reduced soot and NOx emissions in engine design by using the KIVA-GA code in the KIVA-3V CFD model. The start of Injection, Dwell Period, and Mass in the first pulse are kept as a design factor. The model was solved with a Genetic Algorithm framework and the result was compared with the baseline design. Shrivastava *et al.,* [28] used the KIVA-GA code for simulating combustion phenomena, spray characteristics, and emission analysis with a three-dimensional model. They performed thorough engine cyclic simulation and solved it within a Genetic Algorithm framework. They applied gas dynamics codes pertaining to single dimensions for simulating the gas exchange process specifically. Wickman *et al.,* [29] applied nine different configurations as input by making three chamber geometry and six other variables having concerns with chamber geometry at a time. They used the KIVA-GA code for measuring and optimizing engine performance. Brahma and Rutland [30] predicted emission parameters by performing optimization on engine design parameters. They used a neural collaborated network with multi-dimensional CFD code. SAGE model was found helpful in combustion phenomena by combining Chemkin input files and CONVERGE software code [31]. One can include here detailed chemistry involved through the library section. A GT-Suite package has a GT-Power, which is a 1-D tool and it is having one-dimensional gas dynamic as a base. It allows work on heat transfer and flows through the piping model. GT-Power consists of advanced modeling concepts for the internal combustion engine and can be used for simulating steady and transient conditions. Analysis of the vehicle combustion chamber through GT-Power closely agrees with the experimental results [32,33].

For getting the perfect desired mixture of air and fuel as well as to increase this ratio Lin and Ogura [34] have developed an NICS-MH system. The NICS-MH system is used for studying collision amongst fuel particles, impinging diffusion, and effect on wall parameters. NICS-MH system showed improvement in BSFC for high speed and full load. The results of numerical studies from simulation which is known as close cycle simulation showed special agreement to in-cylinder flow characteristics without consideration of intake and exhaust valves [35]. Further investigations show the effective role of turbulence parameters like swirl and tumble in fuel charge mixing quality. Tatschl *et al.,* [36] used AVL FIRE code for simulating the mixing pattern of air-fuel and combustion phenomena and validated their work with experimentation. They used multidimensional simulation for the combustion model. Wieser and Ennemose [37] studied on same configurations for combustion and heat transfer phenomena. Suzzi *et al.,* [38] studied a hybrid model for spray break-up, the study found it competent for spray models of both evaporative and non-evaporative sprays. Higher capacity CI engine tested by Shi and Reitz [39] for getting emission standards within range and to obtain maximum fuel performance at low all-load conditions. They targeted spray pattern, bowl geometry, and swirl as crucial factors for their study. Park [40] achieved permissible emission results and maximum combustion efficiency with baseline geometry. The geometry was further optimized by implementing a deeper cut bowl shape and compared for performance analysis with conventional baseline geometry by using dimethyl ether as fuel. They found notable improvements in performance with permissible emissions value.

The study of Sweetland and Reitz [41] validated the KIVA code to large extent for predicting turbulent kinetic energy. They studied the flow field on KIVA code and used PIV photography in their study by using actual intake port shapes and by keeping a movable design for intake valves. Their study concentrated on the flow field specifically at suction and compression stroke. Erdil and Kodal [42] applied POD i.e., Proper Orthogonal Decomposition, and PAM i.e., Phase Averaging Method to converge the turbulent velocity field in the combustion process of IC Engine. During compression stroke results shows higher velocity values for radial as well as for the circumferential direction. Change in piston bowl geometries seems to be influencing both radial and circumferential components of velocity. Maji *et al.,* [43] achieved a better turbulence profile by getting stronger squish and swirl for re-entrant type chamber geometry. The key parameters set for design were re-entrant diameter, combustion depth, and transition radius. The results obtained in their research were fair for emission standards, particularly for soot and NOx pollutants. Rakapoulos *et al.,* [44] have worked on a study of various piston bowl geometries. They choose the CFD tool and worked on the CFD model for these geometries and the same has been compared with a quasi-dimensional model to figure out results based on pressure and temperature component within the cylinder and velocities in axial and radial components. The result shows CFD model was more promising and acceptable for studying transport phenomena. Models in CFD were made by varying piston bowl diameter to cylinder diameter ratios and with different step sizes. Table 1 shows a summary of numerical studies on the basis of code, model, operating condition, meshing, and experimental.

Table 1
Summarized numerical studies on the basis of code, model, operating condition, meshing and experimental

	Code & Model used in CFD study	Operating Parameters & Set-up	Meshing details	Outcomes	Ref.
1	LES-STAR-CD	DISI Engine motored with four valves	0.6 M cells of coarse grid and 0.15 M cells at TDC having a mean width of 1.2 mm for each cell	TKE value found improved as it leads to refinement of grid structure for in cylinder flow	Chui *et al.,* [45]
2	LES-STAR-CD	DISI Engine 1-S, 4-Cyl., 4-Valves	Polyhedral (6&4), pyramidal, and prismatic grid elements considered. Coarse grid-mean mesh size of 1.2 mm and 0.6 M cells at BDC while fine mesh-0.8 mm mean mesh resolution with 1.5 M cells at BDC	Cyclic variations caused strong different size of vortices with various orientations	Park [40]
3	KIVA	DISI engine 4-S	Hexahedral grid elements 0.41 M cells at TDC & 1.76 M cells at BDC	During compression isotropic state and during intake anisotropic state behavior found	
4	Standard k-e clubbed with PISO	E-TVCS CI-engine N-2400 rpm. Min. valve lift- 0.2 mm, CR-22.6 & 300 K temp. of cylinder wall, port, and valves.	Rectangular and triangular meshes taken for inlet, exhaust and for moving piston conditions	Combustion performance and emissions highly varied by swirl ration and mass average value of TKE found higher in exhaust stroke compared to intake stroke	Hawkes *et al.,* [22]
5	K-e Fluent	Rotary Engine N-8000 rpm, CR-10 and CC-200	115,761 elements of Tetra-hybrid – T grid on GAMBIT	Burning process improved by squish	
6	Fluent RNG k-e	Rotary Engine N-4000 rpm, CR-12.9 and CC-648	Grid size- 2 mm Unstructured tetrahedral cells	Flow breakdown observed during intake to compression stroke	

3.1 Effect of Variation in Shape and Tuning of Intake Ports

All modern automobiles must now meet all the environmental limits as well as the level of fuel efficiency to remain competitive in the global market. Combustion parameters, combustion chamber or piston bowl geometry, compression ratio, fuel pressure in the injector, and fuel purity are all factors that influence engine performance. CI Engine requires proportionate charge which is thought to be the biggest challenge in designing CI Engine. Combustion in IC Engine requires a homogeneous mixture of air and fuel because air motion has an influence on the atomization of fuel and which at last controls the fuel particle distribution for injecting it into the combustion chamber. The key to controlling said parameter is the design and proper orientation of the intake manifold. The shape of the intake manifold has significant and measurable effects on engine performance. The biggest effect which can easily be observed is on the engine's breathing capacity, which impacts the torque and power produced at different engine shaft speeds [10,46]. In order to determine the zone for the peak value of volumetric efficiency in the operating speed range, the intake manifold length is a key regulating factor. Intake manifolds that can provide longer duration for flow results in higher torque at low RPMs and the opposite to those manifolds which provide shorter duration for flow results in peak torque at high RPMs. Some of the researchers have analyzed helical, and spiral, and also combined both shapes for intake manifolds for finding turbulent behaviors. The combined helical and spiral shape when evaluated results in a high swirl and high tumble ratio, effective turbulent kinetic energy, and high volumetric efficiency. As the study shows diesel engine performs better with high swirl flow compared to high volumetric efficiency, so it is better to use a combined helical-spiral manifold for diesel engines. Experiments and simulations on elliptical cross-section intake manifold show larger and stronger tumble motion when compared with circular cross-section intake manifolds [47].

A study was carried out on Biofuel in CI Engine which was initially running on diesel as a fuel but with some modifications as it was to run with Bi-fuel (LPG & Diesel). Results received from simulations and validation of the same with experimental investigations show comparisons between optimal filling geometry intake manifold and initial intake manifold geometry. The result showed improvement in all combustion parameters for modified intake manifold geometry by a minimum of 10 percent. With the same methodology, a diesel engine has experimented with both exhaust valves in close positions and other two inlet valves with a certain lift of around 3-4 mm. The result was carried out for two different geometry of inlet port i.e., tangential and helical. While using a rotating inlet port, it is found no noticeable improvement in turbulence parameters like swirl, tumble, and TKE. Whereas inclined port configuration showed noticeable improvements [14]. Maximum flow charge and flow separation were studied for getting better ideas about a generation of tumble inside the cylinders in different port conditions.

Deshmukh *et al.,* [48] used one dimensional model for the simulation of a 125 cc 4-Stroke SI Engine. The main target of their work was to study the gas interaction process by tuning both outlet and the inlet port. Their simulation analyzed the tuning effect and its effects on engine performance parameters. They optimized results for maximum vehicle speed with improved results for engine performance. Sammut and Alkidas [49] simulated and compared with and without intake & exhaust pipe conditions and noted merits and demerits for both conditions. They tuned the intake port manifold and noted the changes where they found changes in volumetric efficiency because of the rise in pressure inside the cylinder at the closing of the inlet valve and a few more important aspects. They further carried out their study for change in the lengths of intake and exhaust pipes around 1500 rpm. They also studied for changes in valve timing without any changes in valve lift. Vítek and Polášek [50] analyzed the 1-D pipe model for studying the influence of the varying length of the suction manifold on engine parameters. The pipe geometry was compared with a 1300 cc gas engine on a CFD tool and also experimentation was carried out to conclude the results. They observed the effect of changing pipe length with a different configuration of engine speeds and other parameters were kept constant in their studies. The manifold length or pipe length was changed from 500 mm to around 2000 mm whereas speed varied from 1500 rpm to 3000 rpm. Taylor *et al.,* [51] simulated a 1400 cc turbocharged gasoline engine on the GT power CFD simulation tool. They studied the consequence of changing manifold length for said engine. They observed improvement in torque at low speeds and also found improved fuel consumption. The experimental arrangement was capable of changing the length of the intake manifold by keeping inserts of spool pieces. Samuel and Annamalai [52] used AVL BOOST engine simulation software in their research work. They compared results for turbocharged engines and naturally aspirated engines. They concluded that turbocharged engines are more responsive to the variations in the runner length of the intake manifold pipe. They found improvement in engine performance by not just only varying intake manifold length but also along this shaft speed and modified charge system. On the basis of the de-attachment of flow and to avail maximum mass flow thrown away from the port, the exhaust port was modified which results in improved tumble and flow performance compared to conventional design. Table 2 shows a summary of intake port/manifold variation studies on the basis of code, model, operating condition, meshing, boundary constraint, and experimental set-up.

Table 2
Summarized intake port/manifold variation studies on the basis of code, model, operating condition, meshing, boundary constraint and experimental set-up

	Code & Model used in CFD study	Operating Parameters & Set-up	Meshing details	Boundry constraints	Outcomes	Ref.
1	ANSYS Fluent (RNG k-e)	Diesel engine- Multi cylinder, direct injection, 10 mm valve lift & N-1000 rpm	Dynamic mesh- 0.25 M Polyhedral (4 &6) elements	No-slip at wall	Higher swirl ratio found for helical-spiral manifold also higher tumble ratio & higher volumetric efficiency found for helical manifold	Bari and Saad [53]
2	STAR-CD (RNG k-e)	Diesel engine- Single cylinder, direct injection and N-3000 rpm	Hexahedral mesh of 0.3 M cells/liter dense grid for manifold	No-slip at wall & Adiabatic wall	Higher swirl ratio found for helical-spiral manifold also higher TKE & higher volumetric efficiency found for helical manifold	Gugulothu and Reddy [15]
3	ANSYS Fluent (RNG k-e)	Diesel engine- Single cylinder, direct injection, 4-S and N-1500 rpm		No-slip and isothermal wall, Constant pressure at both ports	Greater swirl ratio and higher turbulence found for inclined port configuration compared to rotated port configuration	Firat and Varol [16]
4	STAR-CD (k-e Realizable)	Petrol engine- 3 cylinder and 6 mm valve lift			Flow performance improved by 36 % and tumble improved by 45% by modified inlet and outlet ports	Firat and Varol [16]
5	STAR-CD (RANS	Diesel engine- Single cylinder,	Moving Mesh condition	No slip at wall	Stronger and larger tumble motion observed for elliptical cross section	Galloni [18]

	Standard k-e)	direct injection, 4-S and N-1000 rpm			inlet port compared to circular port	
6	ANSYS (CFX)	Petrol engine- 4 cylinder and 16 valves	Tetrahedral elements (ICEM 10.0)	No-slip at wall and O.2 mm roughness considered	High swirl ratio at high valve timing obtained with compromise in mass flow rate	Ramajo and Nigro [54]
7	Flow Works (Standard k-e)	Diesel engine- 6-cylinder, N-1500 rpm, inlet pressure-1.013 bar		Result obtained for 10 deg and 150 deg crank angles	Increment found for the value of brake-power, torque & BTE at optimal geometry	Liu and Haworth [55]

3.2 Effect of Modifications in Fuel Injection Parameter, Shape of Cylinder Head and Installing Flow Deflectors

Many studies on engine pollution reduction and engine performance enhancement have relied on three key factors: pressure while injecting fuel droplets, time duration for injecting fuel droplets, and compression ratio [56]. The timing of injecting fuel droplets proved to be impactful on combustion and emissions quality [57]. The lower values for initial pressure and temperature were measured when injection time was set as advanced, which resulted in a lagging period for ignition. The NOx emissions values and higher side gas temperature in the cylinder are because of increases in the delay period for ignition [58]. The temperature and pressure are somewhat greater when the fuel injection timing is delayed, reducing the ignition delay interval. Several studies have found that injection parameters influence the combustion and emission parameters of an engine. Rahman *et al.,* [59] worked on experimentation on CI Engine with advanced timing for fuel injection. They carried their observations with various alternative fuels like biodiesels and alcohols. They observed improved emission standards in form of reducing the value of monoxides and hydrocarbon emissions but found a rising value of NOx parameters. They concluded controlling timing for injecting fuel leads to improve brake thermal efficiency besides that they found reducing specific fuel consumption. Study on a single cylinder, compressed ignition, common rail direct injection, diesel engine brings off to run the same configuration on waste cooking oil [60]. The study focused on the effect of modifications in injection timing and injection pressure. During tests by keeping injection pressure values around 80-160 MPa NOx values were measured, and lower NOx emissions were followed by a rise in smoke, according to the findings. Liu *et al.,* [61] used a multi-cylinder (6-cyl) turbocharged and intercooled heavy-duty CI engine to run with diesel/methanol compound combustion mode to conduct experiments. During the advanced injection time, they noted peak values for cylinder pressure and a rise in the values of releasing rates for heat, while BSEC and gas temperature from the exhaust of the engine fell. As diesel injection timings were advanced, NOx levels rose and soot levels fell. Furthermore, due to advancements in injection timing, the HC and monoxide emissions in DMCC mode decreased. Another effective method was port premixing of DME, as well as premixed homogenous air-fuel charge and diffusion combustion on compressed ignition engines as part of the heat release process [62]. When diesel travelled into a cylinder at 7° before TDC, BSFC was lowest at a particular DME quantity. Smog levels decreased, however, CO and HC levels increased as DME levels climbed. With early direct injection, NOx increased during direct injection diesel engine operation, whereas smoke, CO, and HC were reduced. Mean effective pressure and Thermal efficiency increased, whereas Fuel consumption and Gas temperature at exhaust decreased dramatically, according to some researchers [63]. With improved injection timings, emissions of hydrocarbon and monoxide were reduced, but NOx emissions were found on the higher side.

The internal nozzle flow has a significant impact on spray atomization in current diesel engines [64]. While cavitation is undesirable in most engineering processes due to its side effects, such as activities inside the hole of the injectors can improve jet turbulence and therefore the process of atomization of charge in fuel injection systems, notably in current injectors for diesel fuel. Cavitation modeling can be done in one of two ways: using compound or continuum fluid flow models. The fluid flow phases i.e., liquid and vapor are treated separately in the first, but the liquid and vapor phases are treated as a homogeneous mixture in the continuum flow. In these models, cavitation growth can be computed by the application of the equation of state [65].

On the air intake side, the influence of guide vane heights and implantation of components for tumble generation was seen. Guide vane heights of ¼ R, ½ R, and ¾ R (where R is the radii of the inlet runner) were set during research. The study concluded that the larger length of the guide vanes influences more air resistance; thus, ¼ R provided air resistance with a lower intensity value to the airflow. As a result, during the combustion stroke, ¼ R vane profile had higher velocity, tumbled flow, swirl profile, and turbulent kinetic energy. With the transmission of heat aspect, flow fluid pattern, and intensity of turbulence inside the cylinder at a suction stroke, the pent-roof type cylinder outperforms the traditional flat-roof type head [16]. In a mono-cylinder 4-stroke spark-ignited engine, the varied value of tumble ratios caused by tumble deviators affects engine performance [66]. During the study twin-tumble deviators were added in the inlet port to increase tumbling in flow, resulting in enhanced flow velocity and kinetic energy with respect to tumble ratios, as well as the quality speed of flame propagation owing to a qualitative blending of charge. Table 3 here facilities about summarized intake port/manifold variation studies on the basis of code, model, operating condition, meshing, boundary constraint, and experimental set-up.

Table 3
Summarized intake port/manifold variation studies on the basis of code, model, operating condition, meshing, boundary constraint and experimental set-up

	Code & Model used in CFD study	Operating Parameters & Set-up	Meshing details	Outcomes	Ref.
1	ANSYS CFX	Convergence criteria were set at 0.00001	2.75 lacs grid elements- Polyhedral (4 &6)	0.25R cylinder head observed to be most promising for in-cylinder velocity, tumble, swirl, and turbulent kinetic energy	Huang [8]
2	CONVERGE (k-e with high Re)	Tumble ratios set on 0.5, 1.5 & 2.2, Grid size set on 3mm and tested for 30 cycles		With increasing value of tumble ratio values for flow velocity and TKE increases which improves flame propagation speed and mean effective pressure due to better air-fuel mixing	Gugulothu and Reddy [15]
3	2D Standard k-e N-S Equation	By neglecting buoyancy forces		Higher in cylinder turbulence value found in pent-roof type cylinder head compared to the conventional flat-roof type head	Ramajo *et al.,* [24]
4	ANSYS Fluent Standard k-e	Diesel engine- 4-S and N-2400 rpm	Hexagonal mesh	The SOI time of 18 deg BTDC found to be best for most effective results	Treurniet *et al.,* [21]

3.3 Effect of Varying Crown or Bowl Shape

Squish and swirl components of any turbulent flow help to improve the air-fuel mixture in general. At the premix stage or before ignition if somehow one can develop squish and swirl proper mixing at this stage can be achieved and that can improve combustion characteristics [67]. The geometry formed by the cylinder head and top surface of the piston also called as piston crown or piston bowl has an impact on a generation of squish and swirl which in the end is proven to improve ignition characteristics. Piston bowl geometry thus strongly has an impact on combustion and also it can improve combustion performance parameters as well as emission parameters. Harshvardhan and Mallikarjuna [68] concluded that better performance in terms of tumble ratio and turbulent kinetic energy was reported for the flat shape of the piston head compared to flat piston center bowl geometry and opposite to that flat piston bowl shape provides better flow structures while comparing it with dome piston bowl shape during their experimental work and CFD simulation. Further, they concluded that the fuel evaporation rate increased while using a flat piston head shape. Gugulothu and Reddy [15] studied flat piston bowl shape by experimenting on CI engine and getting validated the same with simulation on CFD and noted that flat piston bowl shape produces higher pressure inside the engine cylinder which directs higher values for energy during suction and compression strokes. Further, they found higher values for brake power and swirl ratio with the usage of a flat bowl shape. The offset bowl shape reported the highest turbulent kinetic energy in that experiment. Further, they compared modified pistons on the basis of toroidal diameter, throat diameter, and shapes like a piston with a center pit and flat bowl &piston with high throat value, they found toroidal kind of pistons gave better results on the basis of swirl and tumble aspects compared to others. Bibu and Rajan [17] noted the effect of the shape of the piston bowl and the motion of the piston on the location of the tumble vortex. They reported double-lobed piston increases by almost 70 % swirl ratio during suction and 92% during compression stroke while having a comparison with a flat piston bowl shape. Nicollet *et al.,* [69] studied the importance of annular flow with respect to the motion of the piston and sown velocity distribution for the same.

Jaichandar and Annamalai [70,71] in their study used four different profiles for piston bowl geometries. They identified the impact of changes in geometries on performance parameters by using biofuel on CI diesel engines. They fixed the start of injection, swirl ratio, and fraction of fuel injected as a variable for optimizing light-duty diesel engines. They used two spray angles of a nozzle with a different configurations of piston bowl shapes. Raj *et al.,* [72] optimized different configurations of flat, inclined, center bowl, and inclined offset piston top surfaces on CI engines for combustion characteristics and air stream mixing. Mobasheri and Peng [73] concluded that lowering the dimensions of bowl depth on the piston head surface results in higher values of NOx. They conducted experiments on a high-speed diesel engine. They studied the effect of Re-entrant shape chamber geometry both on CFD and by experimentation and found the shape of the piston bowl influences the performance parameters of an engine.

Kidoguchi *et al.,* [74] proved that emission elements such as particulate matter and NOx, which are detrimental to the environment and human health, can be reduced by lowering throat diameter and by increasing the magnitude of squish by some means. Subramanian *et al.,* [75] worked on a mono-cylinder diesel engine, and also simulated the same on CFD. They validate their work with experimentation and concluded that the generation of turbulence and swirl has a huge impact on combustion phenomena. They noted combustion bowl shapes are impactful for the generation of turbulence and swirl which in turn increases turbulent kinetic energy. They experimented with three different configurations in the shapes of combustion chamber profiles for CI engines. The engine used was in an application for agriculture usage. They tested the engine for getting results for performance and airflow characteristics. They optimized various parameters of performance and emissions like the viscosity of turbulent flow, kinetic energy during turbulence, mixing of air and fuel, fuel consumption, NOx, smoke emissions, and stages of combustion and diffusion. They found improved said parameters during their studies. Gafoor and Gupta [76] numerically examined a CI engine with different swirl ratios. They found the effect of enhancing turbulence on combustion and emission parameters. They proved piston swirl with a high bore-to-bowl ratio configuration has a huge effect on pressure aroused inside the cylinder and also has an effect on soot and NOx emissions. Prasad *et al.,* [77] examined the consequences of modified shapes of bowl geometry, mainly re-entrant type on soot formation, NOx formation, and other emissions as well as performance parameters. Their results concluded that re-entrant geometry produces higher swirl and thus better turbulence profile which improves combustion and emission parameters. They studied the behaviors of CI engines by varying different configurations of piston bowl geometry. They tested and compared the engine for performance and emissions analysis. They found increased toroidal radius proves better for increased combustion efficiency and reduced soot & particulate matter. The re-entrant geometry shape was found to be accurate for direct injection CI engines due to the high-pressure generation inside the cylinder and precise soot reduction process. Many other researchers worked on re-entrant geometry shapes of piston bowl profiles and optimized the shape of the profile through simulations on CFD by varying controlled parameters like throat diameter, toroidal radius, and others as discussed earlier. The main motto of their research involved reducing soot & particulate matter, like emission parameters, and for the increased performance parameters by working on combustion performance parameters. Table 4 provides summarized piston crown shape variation studies for various CFD codes, parameters, and analytical results.

Table 4
Summarized Piston Crown Shape Variation studies for various CFD codes, parameters, and analytical results

	Code & Model used in CFD study	Operating Parameters & Set-up	Meshing & Boundry constraints	Outcomes	Ref.
1	STAR-CD (Standard k-e)	Diesel engine-Single cylinder, 2-valve and 4-S		For higher tumble ratio and TKE, Flat-piston head found better than flat-piston center-bowl	da Costa *et al.,* [78]
2	STAR-CD (RNG k-e)	Diesel engine-Single cylinder, 2-valve and 4-S	Polyhedral trimmed cells and 10 holes considered for fuel injector	At injection and ignition conditions the flat piston with center bowl gives higher in-cylinder TR and TKE when compared with piston with no bowl	Azad *et al.,* [79]
3	Standard k-e	Diesel engine-Single cylinder, 2-valve and 4-S	No-slip conditions and orthogonality and skewness checked for mess	Highest engine pressure resulted on flat piston bowl geometry	Toh *et al.,* [47]
4	Standard k-e	Petrol engine-Direct injection type		Enhanced swirl and tumble inside cylinder achieved by Piston modified into toroidal diameter with center pit has more effect compared to flat bowl and flat piston with higher throat diameter	Ramajo *et al.,* [24]
5	ANSYS Fluent standard k-e		Hexagonal and tetragonal elements for	The swirl ratio increased 66.67% for intake stroke and 91.47% for	Vu and Guibert [26]

			moving and static conditions	compression stroke for double-lobed piston head	
6	LES	DISI engine-Optical	Meshing generated on CENTAUR	The Location of tumble vortex affected by the piston motion and piston bowl shape	Brahma and Rutland [30]

3.4 Effect of Valve Lift, Engine Speed and Crank Angle Variation

A camshaft (or camshafts) during its rotation which is half the speed of the crankshaft controls valve movement. The crankshaft revolves twice during the four strokes cycle, creating two sets of piston movement cycles, and the camshaft thus rotates once, causing one valve cycle. When defining the time for valve opening & shutting with angles, the various speeds of the crank & cam might cause some misunderstanding, as full one rotation i.e., 360° of crankshaft rotation is comparable to half the circle i.e.,180° of camshaft revolution. The valve timing parameter is usually measured by taking reference to the piston position. It is the reference position of the piston by measuring the angle of the crankshaft for piston motion between TDC to BDC. Valve timing duration is the value of the angle for crankshaft rotation between opening and closing either taking the inlet valve or exhaust valve as a reference. As per works of literature as soon as the exhaust valve opens up pressure is released from the cylinder because of combustion to the exhaust system as an escape. So, for having maximum gettable work or to achieve higher efficiency, it is undesirable to open the exhaust valve before the piston approaches BDC. Further, exhaust back pressure values (blow-down pumping work) are intended to be at the lower side before the piston approaches towards upward motion. The above discussion arouses two conflicting requirements i.e., Exhaust Valve Opening after BDC and Exhaust Valve Opening before BDC. The decision is always adjustment between the work lost and the work required to move the piston whilst the cylinder pressure is still above the exhaust back-pressure and by allowing the combusted gas to escape before it is fully expanded.

The timing for exhaust valve opening (EVO) has to be optimized by knowing other effects like engine speed, load on the crankshaft, and pressure of the gas inside a cylinder. Study shows that at part load EVO has to be moved closest possible to BDC to achieve maximum output. This is because in this situation pressure inside the cylinder is nearly the same as exhaust back pressure which at that instant takes lesser time to leave from the valve end.

Likewise, Exhaust Valve Closing (EVC) is also an important parameter, it has a significant effect on exhaust gas as a residue just before the beginning of a suction stroke. EVC with respect to EVO defines valve overlap and is useful to know the effects at beginning of suction. Just after the exhaust stroke and commencement of suction stroke for operating at full load conditions, it is always desirable minimum exhaust quantity remains as a residue. This will allow maximum fresh charge to go for suction. This requires the exhaust valve to be closed within the shortest time after the piston approaches TDC. The duration of EVC influences the direction of pressure waves due to exhaust gas which in turn decides flow that either draws gas outwards or flow that pushes gas back towards the cylinder as a residue. So, the timing for EVC has to be optimized for different speeds of the engine to regulate pressure waves from the exhaust gas.

During partial loading operation, it is always advisable to hold on to some amount of exhaust gas inside a cylinder. The residue reduces the need for a throttle plate and thus reduces pumping loss at the intake stroke. Retain gas itself restricts fresh charge to enter and works as a throttle valve partly.

Exhaust gas recovery (EGR) has a certain effect on stability for combustion. It has a dependency on cylinder volume in such a manner that quantity when it is in tolerable proportion can continue stable combustion. However, lowering engine load and reduction in charge density simultaneously happens during the supply of exhaust gas through EGR which shows the tendency of the combustion process means an increase in the value or quantity of EGR affects reduction in combustion rate or slow combustion process which further makes the process of combustion in unstable range.

The timing for the opening of the inlet valve which in short is also known as IVO is also an important parameter that needs to be studied. IVO affects valve overlapping. An Inlet valve opening before the piston approaches the uppermost position may result in the backflow of gases toward the intake manifold as a residue.

The timing for closing of the inlet valve which in short is also known as IVC affects the volumetric efficiency of an engine. Whatever charge is sucked inside the cylinder during intake stroke is trapped by proper closing of the inlet valve. The presence of a pressure wave at the inlet valve side usually resulted in airflow toward the cylinder after the piston approaches BDC. Here also it should be clear that optimum timing for IVC varies according to engine speed variations. As this parameter increases timing for IVC shift towards after BDC to get more advantage from pressure waves. Timing either advancement or delay for the closing of the inlet valve to obtain maximum torque value may result in lower air mass captured in the cylinder. Advancement or early shutting reduces airflow capacity to get into the cylinder and retardment or late closing of IVC may cause a reverse flow of air toward the intake manifold. IVC timing is an important factor for volumetric efficiency and further improves engine performance and emission parameters.

Valve overlap may happen when both the suction as well as exhaust valves are open at the same instance. At any engine speed and any load valve overlapping creates an opportunity for intake and exhaust flow to influence each other. In an ideal condition, the overlapping is meant to avail the cylinder with a fresh charge at intake without bypassing the path of fresh air directly toward the exhaust side. The amount of intake charge is estimated to increase the amount of exhaust gas that could be deduced into the cylinder by the swept volume of the piston solely, in the combustion chamber at the top dead.

The amount and duration of valve lift is a crucial consideration to take into account when researching engine performance, as the peak lift of the valve directly controls airflow towards intake and exhaust. Valve lift haves a practical limitation for its peak value based on the design of the specific engines. During the design of the piston, the peak values for valve lift must not unduly compromise as usually the top surface of the piston which is also known as the crown used to be profiled for clearance. The higher the engine running speed more the value for valve lift at a peak is restricted for a certain duration. The lower side value of the valve lift restricts airflow both ways i.e., towards the engine cylinder and outwards from the exhaust side. For getting maximum torque at the shaft and given engine speed valve lift should be kept as less as feasible. Further, to achieve maximum energy power as an output at a given engine speed valve lift should be as more as feasible.

Because of tumble decomposition, TKE has the best amount all through suction and at the end of the compression stroke close to top-dead, and tumble momentum is related to engine speed [18]. Sizable swirls in a rotary Wankel engine change to tiny tumbles as the inlet angle, inlet pressure, and rotary speed increase [19]. When the rotational speed and intake pressure are increased, the average TKE increases, but when the crank angle is increased, TKE reaches a maximum value. The RNG k-Ɛ model also produces quite precise in-cylinder flow measurements than the basic and realizable k-Ɛ models. For low-rpm computations, however, the standard k-Ɛ model is recommended [80]. TKE and tumble ratio enhances as engine speed increases, but as the compression ratio is greater, TKE declines while the tumbling ratio increases. Table 5 shows a Summarized study of variations in crank angle, engine speed, and valve lift, for different CFD codes, parameters, and analytical results.

Table 5
Summarized study of variation in crank angle, engine speed and valve lift, for different CFD codes, parameters, and analytical results

	Code & Model used in CFD study	Operating Condition	Outcomes	Ref.
1	STAR-CD (RNG k-e)	GDI engine- Single cylinder, 2-S. N-500, 1000 and 1500 rpm & CR-6,7	With increase in engine speed TKE and tumble ratio increases whereas with rise in compression ratio TKE rises and tumble ratio falls	Zhang *et al.,* [81]
2	ANSYS-Fluent (Realizable k-e)	Single cylinder engine at N-800 rpm.	Higher flow velocity, turbulence, and swirl intensity achieved by little valve lifts	Chen *et al.,* [82]
3	ANSYS (ICEM)	Fiat petrol engine-4 cylinder, 16 valves and N- 1500, 3000 and 4500 rpm	More swirl is created by valve lifts but the volumetric efficiency drops due to it	Nishad *et al.,* [83]
4	ANSYS Fluent (RNG k-e)	N-2000, 4000, 6000 and 8000 rpm	As rotating speed and intake pressure increases TKE grows further	Nicollet *et al.,* [69]
5	CONVERGE (RNG k-e)	Petrol engine- Single cylinder, 4 valves, CC-125and Air (300K & 1Bar) as working fluid	Rise in valve lift and pressure drop results into increase in the magnitude of flow velocity and vorticity	Treurniet *et al.,* [21]

4. Comparative Study of Turbulence Models

In any fluid flow problem, the fundamental energy equation also known as Naiver-Stokes equation is governing constants and variables pertaining to mathematics for describing exchanges that happen in mass, momentum, and energy transmission. CFD solves partial differential equations that describe flow analytically to model any fluid flow problem. CFD, for the most part, predicts turbulent flow using one of the three turbulence models: (i) Direct Numerical Simulation (DNS), (ii) Large Eddy Simulation (LES), and (iii) Reynolds Averaged Naiver Stokes (RANS). The RANS and LES methodologies are combined in the DES turbulence model. The primary idea of DES is to use the RANS model with due consideration of boundary layers and the LES model in the separation area, which reduces computational resources while ensuring accuracy. To compute turbulent flow in DNS, the fundamental energy equation is used without the consideration of any kind of approximation. DNS requires excellent grid resolution to locate the tiniest eddies formed particles in a turbulent flow, which is why a huge computer is required. The work of LES focuses on the enlarged view structure of the turbulent flow. Turbulence can be divided into big and small eddies for LES analysis, and large eddies are taken into account in the simulation. Existing computational capabilities allow for the solution of 3-dimensional and time-dependent equations for massive eddies. The LES method of turbulence simulation is widely employed. LES is based on self-similarity theory and simulates small vortices using a subgrid scale, whereas large vortices are calculated geometrically. The disadvantage of this model is that calculating the region near the wall is difficult. Large-scale vortices dominate the LES simulation method's transmission of fluid velocity, mass, energy, and other physical quantities. Geometric and boundary conditions have no bearing on small-scale vortices with isotropic motion. To calculate turbulent motion bigger than the grid scale, LES employs the N-S equation. In the Fourier equation or spatial domain N-S equation, the LES model derives a filtered momentum equation by filtering out vortices smaller than the filtered grid. The core idea behind RANS is to use temporal averaging on the N-S equations to turn an unstable complex turbulence kind of problem into a steady kind of problem, at the additional cost involved for unknown numbers that are also in the form of stress, called Reynold's stress. A turbulence model is also required to characterize Reynold's stress. The information given in the equation has been somewhat lost due to the problem's time averaging.

Giving a Reynolds stress model and applying it to all flows is very challenging. For modeling turbulent flow, the Reynolds- Averaged Naiver–Stokes (RANS) method is used rather than using prompt flow parameters. By calculating the average of both static and dynamic flow variables, RANS can mimic turbulence in a flow. Several researchers used computational fluid dynamics (CFD) to compare and select the optimal turbulent parameters for flow characterization under various operating situations in a cylinder. LES and RANS model were compared by using STAR-CD code for the swirl vanes around the valve stem [79]. Result agreed more than 80% for computing total TKE at certain meshing conditions. Also, while making comparisons LES model shows significant variations with respect to the RANS model for one to another cycle. As the RANS model average out vector profiles for velocity for more than one cycle and due to that in terms of cyclic changes, LES produces more realistic findings. Both sub-filter and mesh models show an impact on flow results. Detached Eddy Simulation (DES) using a conventional k-Ɛ turbulence prototype was numerically compared to large eddy simulations with the LES-RANS hybrid [47]. LES had higher ratios for swirl & tumble with respect to RANS & DES for the whole cycle throughout. RANS, DES, and LES solutions had identical mean velocity profiles, but DES had significantly lower cycle-to-cycle variability. Cold flow analysis produces extremely more variations for turbulent cycles, and CCVs can be boosted by adding engine ignition conditions. Three different approaches to large-eddy simulation are also examined. Due to its moderate grid resolution, the Smagorinsky model outperforms total LES and scale selective discretization [84]. Residual turbulence in the inlet zone has a significant impact on cyclic fluctuation throughout the cylinder. On the same track, the Delayed Detached Eddy Simulation (DDES) model is very reliable and compels with respect to the RANS and LES models within an axisymmetric rapid expansion geometry. For the LES-considered section in the DES model, the refining process for the grid also helped to achieve greater resolutions resulting in the flow structures and turbulence attributes. Inside the flow at the inlet port, the Sigma turbulence model, or LES model, was quantitatively scale-up besides the Detached Eddy Simulation-Shear Stress Transport (DES-SST) [85,86]. The Sigma model with a highly precise meshing had the idealistic resulted concurrence compared to the results from PIV, but the DES model had the most variance in the highest velocity and its profiles. In terms of mean values for radial & upward velocities component, length ratio, and tumble, the results of the fundamental RNG k-Ɛ model are closely confirmed according to experimentation with respect to the basic k-Ɛ model during intake and compression [24]. The RNG k-Ɛ model is producing more realistic outcomes than the regular k-Ɛ and other modified k-Ɛ models for flow behaviors (ranging less than 8-10% error). RNG k-Ɛ producing results are matching the experimental mean velocity magnitude better with respect to k- ω SST [82]. k- ω, on the other hand, outperforms the k-Ɛ model in terms of predicting TKE and qualitative scalar velocity fields. LES's peak TKE has greater maximum values than DNS's. Both the LES model & DNS model's simulations were concurrent with the experimental data on cyclic variability, and they produced roughly equivalent conclusions. The RANS k-Ɛ model was shown to have low accuracy in normalizing velocity vectors when compared to the LES [83]. The in-cylinder turbulence was caused by mainly due to the structures followed by turbulence that found formed along with the valve port. Throughout the previous decade, the LES and k-Ɛ models were found applied often for numerical computations for flow fields in a cylinder. This is due to more precise and accurate estimating and findings that come out while making a comparison with other turbulence models. Some models for studying turbulence phenomena, such as k- ω or realizable k-Ɛ, are rarely employed for computing fluid movement since it is unsuitable for predicting in-cylinder flow field behavior, which results in inaccurate and unreliable findings. Although IC engines are becoming less common around the world, the number of research projects involving numerical analysis for studying flow patterns and flow characteristics for flow in a cylinder is still large, and it appears to remain consistent. Table 6 gives a summary based on turbulence Models Comparison for different CFD codes, parameters, and analytical results.

Table 6
Summarized study on the basis of turbulence Models Comparison for different CFD codes, parameters, and analytical results

	Code & Model used in CFD study	Operating Condition	Meshing details/Boundary conditions	Outcomes	Ref.
1	STAR-CD LES (RANS k-e)	An axisymmetric piston-cylinder setup, N-200 rpm and Air as a working fluid	Coarse mesh. No slip & Adiabatic wall for hexahedral grid elements	Considerable cyclic fluctuations observed for LES compared to RANS	Liu *et al.,* [64]
2	STAR-CD LES & DES (RANS standard k-e)	Diesel engine- Single cylinder, 4 valves, N-1500 rpm and injection of n-heptane twice a cycle	Mesh of 0.71 mm average resolution considered	Throughout the cycles, Higher swirl and tumble ratios achieved by LES with respect to RANS and DES. Mean velocity profiles observed identical for LES, RANS and DES	di Mare *et al.,* [87]
3	Standard & RNG k-e	Diesel engine-N-1200 rpm, 295 K intake air temp. and 0.51 bar cylinder pressure		Experimental evidence shows that in comparison to the traditional k-e model, the RNG k-e model's results found more consistent	Baum *et al.,* [88]
4	STAR-CD standard k-e, RNG k-e & chen k-e	N-1500 rpm		When compared to other models in terms of in-cylinder flow fields and characteristics (< 8% error) the RNG k-e turbulence model gives accurate results	Baumann *et al.,* [89]
5	STAR-CD, ICE (RNG k-e) & SST (k-x)	N-1000, 1500 and 2000 rpm speed	Adiabatic wall considered	Experimental mean velocity magnitude curve found better in The RNG k-e results while compared to the k-x SST values	Hawkes *et al.,* [20]
6	Open-FOAM LES	DISI engine with high Reynolds number consideration	Hexahedral mesh	In comparison to the LES, the RANS k-e model was found to be significantly less accurate in normalizing velocity vectors	Anand *et al.,* [90]
7	Implicit LES & scale selective discretization	N-200 rpm, 60 mm stroke and 4 mm valve lift		Because of its moderate grid resolution, the Smagorinsky model outperformed DNS and measurements	Wickman *et al.,* [29]
8	LES & DNS	DISI engine- N-2000 and 3000 rpm speed. 10 cycles tested to represent the cyclic variations		By increasing engine speed, the average RFKE rose as well. Significant turbulent CCVs are produced via cold flow study	
9	Open-FOAM DES, RANS & LES		Polyhedral cells chosen arbitrarily	DDES turbulence model used outperformed the RANS model in terms of accuracy and precision	
10	LES & DES-SST	298 K temp.,3 kg/min mass flow rate and valve lift at 5 mm	Polyhedral elements (4 & 6)	Results of PIV matched by sigma model with fine mesh, whereas DES had the most variance in maximum velocity	

5. Conclusion

Based on engine specifications and operating conditions, distinct methodologies for study flow behaviour are carried out employing using different models and codes in CFD software. For in-cylinder flow analysis, most of the utilized codes may be from the following software like ANSYS Fluent, CFX or ICEM modules, KIVA, Open FOAM, and CONVERGE. When compared to experimental results, all of these codes yield significantly more accurate results. Because of the non-deterministic nature of turbulence, simplifications and assumptions have been permitted for the modelling of fuel charge spray patterns, combustion, and turbulence. Moreover, uncertainties occur because of a consequence of boundary conditions like temperature gradient at the cylinder wall and also variations in many other parameters caused by timing and delay period of ignition and injection event. The review confirms that the standard k- Ɛ turbulence model and the RNG k-Ɛ turbulence model both can assure close outcomes by simulating the fluid flow for convective heat transfer case irrespective of forced or mixed convection phenomenon. The RNG k- Ɛ model solves problems that are closer to the forced convective heat transfer case. During simulation tolerance of the meshing scheme and time steps on the standard k- Ɛ model is better than the other k- Ɛ model, and the overall effectiveness of the standard k- Ɛ model is better than the other k- Ɛ model. The standard k- Ɛ model, which generates similar outcomes to the experiments, is widely used, but there is stronger and more precise k-Ɛ model available. The renormalized k- Ɛ model outperforms the standard and chemkin k-Ɛ models because it generates results near the wall boundary, it should be used in such circumstances.

Large Eddy Simulations solve Navier-Stokes equations directly, creating a relatively accurate turbulence model. Besides that, whenever the flow pattern forms sizable tumble structures, LES is best suited during the intake stroke. Because the large tumble collapses into tiny particles during the compression stroke, LES yields substandard outcomes. CFD simulations forecasts and give deep insights into the flow pattern persisting in different conditions like turbulence, in-cylinder flow, mixture formation, combustion, and emission that pure experiments could not deliver conveniently or economically, so contributions are frequently requested to specific combustion or emission models while accepting model simplifications such as turbulence and spray. As an outcome, according to the requirement of complex chemical mechanisms and complex combustion phenomena, models and codes that account for the complex mixing of different fuels are expected to be developed. As a result, soon, a variety of such codes and models will be needed to be available commercially to prove their specific capabilities concerning the desired combustion process and to strengthen the engine development process with CFD.

References

[1] Hayder, G., and P. Puniyarasen. "Identification and evaluation of wastes from biodiesel production process." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 3, no. 1 (2016): 21-29.

[2] Patel, Nikul K., Anand K. Patel, Ragesh G. Kapadia, and Shailesh N. Shah. "Comparative study of production and performance of bio-fuel obtained from different non-edible plant oils." *International Journal of Energy Engineering* 5, no. 3 (2015): 41-47.

[3] Jamil, Abdullah, Masri B. Baharom, and A. Rashid A. Aziz. "IC engine in-cylinder cold-flow analysis-A critical review." *Alexandria Engineering Journal* 60, no. 3 (2021): 2921-2945. https://doi.org/10.1016/j.aej.2021.01.040

[4] Cupiał, K. A. "Comparison of Gas Engines with Different Combustion Systems." In *First International Conference on Gas Engines, Motortech*, pp. 7-13. 2003.

[5] Sarwani, Muhamad Khairul Ilman, Mas Fawzi, Shahrul Azmir Osman, Anwar Syahmi, and Wira Jazair Yahya. "Calculation of Specific Exhaust Emissions of Compression Ignition Engine Fueled by Palm Biodiesel Blend." *Journal of Advanced Research in Applied Sciences and Engineering Technology* 27, no. 1 (2022): 92-96. https://doi.org/10.37934/araset.27.1.9296

[6] Patel, Nikul K., Ragesh G. Kapadia, Romit Y. Gandhi, and Shailesh N. Shah. "Engine performance, emission and fluorescence study of fuel from plant oils." *Journal of Mechanical Engineering Research and Developments* 40, no. 4 (2017): 673-691.

[7] Shekh, Asfakahemad, and Nikul Patel. "Impact of compression ratio & injection pressure on NOx emission behaviour of VCR diesel engine powered by Madhuca Longifolia biodiesel." *European Chemical Bulletin* 12, no. 5 (2023): 1106-1127.

[8] Huang, R. F., K. H. Lin, C-N. Yeh, and J. Lan. "In-cylinder tumble flows and performance of a motorcycle engine with circular and elliptic intake ports." *Experiments in Fluids* 46 (2009): 165-179. https://doi.org/10.1007/s00348-008-0551-z

[9] Kapitza, L., O. Imberdis, H. P. Bensler, J. Willand, and D. Thévenin. "An experimental analysis of the turbulent structures generated by the intake port of a DISI-engine." *Experiments in Fluids* 48 (2010): 265-280. https://doi.org/10.1007/s00348-009-0736-0

[10] Paul, Benny, and V. Ganesan. "Flow field development in a direct injection diesel engine with different manifolds." *International Journal of Engineering, Science and Technology* 2, no. 1 (2010): 80-91. https://doi.org/10.4314/ijest.v2i1.59089

[11] Jaat, Norrizam, Amir Khalid, Norrizal Mustaffa, Fathul Hakim Zulkifli, Norshuhaila Mohamed Sunar, Ridwan Saputra Nursal, Mahmod Abd Hakim Mohamad, and Djamal Didane. "Analysis of injection pressure and high ambient density of biodiesel spray using computational fluid dynamics." *CFD Letters* 11, no. 1 (2019): 28-39.

[12] Maksom, Mohammad Syahadan, Nurul Fitriah Nasir, Norzelawati Asmuin, Muhammad Faqhrurrazi Abd Rahman, and Riyadhthusollehan Khairulfuaad. "Biodiesel composition effects on density and viscosity of diesel-biodiesel blend: a CFD study." *CFD Letters* 12, no. 4 (2020): 100-109. https://doi.org/10.37934/cfdl.12.4.100109

[13] Niknahad, Ali, and Abdolamir Bak Khoshnevis. "Numerical study and comparison of turbulent parameters of simple, triangular, and circular vortex generators equipped airfoil model." *Journal of Advanced Research in Numerical Heat Transfer* 8, no. 1 (2022): 1-18.

[14] Kaplan, Mahmut, Mustafa Özbey, and Hakan Özcan. "Numerical Investigation of the Effects of Intake Port Geometry on In-Cylinder Motion and Combustion in Diesel Engine." *The International Journal of Engineering and Science* 7, no. 6 (2018): 16-26.

[15] Gugulothu, S. K., and K. H. C. Reddy. "CFD simulation of in-cylinder flow on different piston bowl geometries in a DI diesel engine." *Journal of Applied Fluid Mechanics* 9, no. 3 (2016): 1147-1155. https://doi.org/10.18869/acadpub.jafm.68.228.24397

[16] Firat, Müjdat, and Yasin Varol. "A comparative analysis of in-cylinder flow, heat transfer and turbulence characteristics in different type combustion chamber." *International Journal of Automotive Engineering and Technologies* 7, no. 1 (2018): 18-28. https://doi.org/10.18245/ijaet.438043

[17] Bibu, B., and Vikas Rajan. "Numerical simulation of cold flow analysis of internal combustion engine with double-lobed piston head." In *Advances in Fluid and Thermal Engineering: Select Proceedings of FLAME 2018*, pp. 657-668. Springer Singapore, 2019. https://doi.org/10.1007/978-981-13-6416-7_61

[18] Galloni, Enzo. "Analyses about parameters that affect cyclic variation in a spark ignition engine." *Applied Thermal Engineering* 29, no. 5-6 (2009): 1131-1137. https://doi.org/10.1016/j.applthermaleng.2008.06.001

[19] Fan, Baowei, Jianfeng Pan, Aikun Tang, Zhenhua Pan, Yuejin Zhu, and Hong Xue. "Experimental and numerical investigation of the fluid flow in a side-ported rotary engine." *Energy Conversion and Management* 95 (2015): 385-397. https://doi.org/10.1016/j.enconman.2015.02.047

[20] Hawkes, Evatt R., Ramanan Sankaran, James C. Sutherland, and Jacqueline H. Chen. "Direct numerical simulation of turbulent combustion: fundamental insights towards predictive models." In *Journal of Physics: Conference Series*, vol. 16, no. 1, p. 65. IOP Publishing, 2005. https://doi.org/10.1088/1742-6596/16/1/009

[21] Treurniet, T. C., F. T. M. Nieuwstadt, and B. J. Boersma. "Direct numerical simulation of homogeneous turbulence in combination with premixed combustion at low Mach number modelled by the-equation." *Journal of Fluid Mechanics* 565 (2006): 25-62. https://doi.org/10.1017/S0022112006002072

[22] Lignell, D. O., Jacqueline H. Chen, and Edward Simon Richardson. "Terascale direct numerical simulations of turbulent combustion-fundamental understanding towards predictive models." In *Journal of Physics: Conference Series*, vol. 125, no. 1, p. 012031. IOP Publishing, 2008. https://doi.org/10.1088/1742-6596/125/1/012031

[23] Zulkurnai, Fatin Farhanah, Norhidayah Mat Taib, Wan Mohd Faizal Wan Mahmood, and Mohd Radzi Abu Mansor. "Combustion characteristics of diesel and ethanol fuel in reactivity controlled compression ignition engine." *Journal of Advanced Research in Numerical Heat Transfer* 2, no. 1 (2020): 1-13.

[24] Ramajo, Damian Enrique, Angel Luis Zanotti, and Norberto Marcelo Nigro. "In-cylinder flow control in a four-valve spark ignition engine: numerical and experimental steady rig tests." *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering* 225, no. 6 (2011): 813-828. https://doi.org/10.1177/0954407011400153

[25] Chen, Hao, David L. Reuss, and Volker Sick. "On the use and interpretation of proper orthogonal decomposition of in-cylinder engine flows." *Measurement Science and Technology* 23, no. 8 (2012): 085302. https://doi.org/10.1088/0957-0233/23/8/085302
[26] Vu, Trung-Thanh, and Philippe Guibert. "Proper orthogonal decomposition analysis for cycle-to-cycle variations of engine flow. Effect of a control device in an inlet pipe." *Experiments in Fluids* 52 (2012): 1519-1532. https://doi.org/10.1007/s00348-012-1268-6
[27] Park, Sungwook. "Optimization of combustion chamber geometry and engine operating conditions for compression ignition engines fueled with dimethyl ether." *Fuel* 97 (2012): 61-71. https://doi.org/10.1016/j.fuel.2012.03.004
[28] Shrivastava, Rahul, R. Hessel, and Rolf D. Reitz. "CFD optimization of DI diesel engine performance and emissions using variable intake valve actuation with boost pressure, EGR and multiple injections." *SAE Transactions* (2002): 1612-1629. https://doi.org/10.4271/2002-01-0959
[29] Wickman, David D., Peter K. Senecal, and Rolf D. Reitz. "Diesel engine combustion chamber geometry optimization using genetic algorithms and multi-dimensional spray and combustion modeling." *SAE Transactions* (2001): 487-507. https://doi.org/10.4271/2001-01-0547
[30] Brahma, Indranil, and Christopher J. Rutland. "Optimization of diesel engine operating parameters using neural networks." *SAE Transactions* (2003): 2521-2529. https://doi.org/10.4271/2003-01-3228
[31] Senecal, P. K., E. Pomraning, K. J. Richards, T. E. Briggs, C. Y. Choi, R. M. McDavid, and M. A. Patterson. "Multi-dimensional modeling of direct-injection diesel spray liquid length and flame lift-off length using CFD and parallel detailed chemistry." *SAE Transactions* (2003): 1331-1351. https://doi.org/10.4271/2003-01-1043
[32] Guo, Chendong, Yu Song, Huihua Feng, Zhengxing Zuo, Boru Jia, Ziwei Zhang, and A. P. Roskilly. "Effect of fuel injection characteristics on the performance of a free-piston diesel engine linear generator: CFD simulation and experimental results." *Energy Conversion and Management* 160 (2018): 302-312. https://doi.org/10.1016/j.enconman.2018.01.052
[33] Božić, Mladen, Ante Vučetić, Momir Sjerić, Darko Kozarac, and Zoran Lulić. "Experimental study on knock sources in spark ignition engine with exhaust gas recirculation." *Energy Conversion and Management* 165 (2018): 35-44. https://doi.org/10.1016/j.enconman.2018.03.053
[34] Lin, Bai-fu, and Masaru Ogura. *A New Multi-impingement-Wall Head Diffusion Combustion System (NICS-MH) of a DI Diesel Engine - The Effect of Combustion Chamber Geometry*. No. 951792. SAE Technical Paper, 1995. https://doi.org/10.4271/951792
[35] Payri, F., J. Benajes, X. Margot, and A. Gil. "CFD modeling of the in-cylinder flow in direct-injection Diesel engines." *Computers & Fluids* 33, no. 8 (2004): 995-1021. https://doi.org/10.1016/j.compfluid.2003.09.003
[36] Tatschl, R., H. P. Gabriel, and P. Priesching. "FIRE-A Generic CFD Platform for DI Diesel Engine Mixture Formation and Combustion Simulation." In *International Multidimensional Modeling User's Group Meeting at the SAE Congress*, Detroit, Mich., USA. 2001.
[37] Wieser, K. J., and A. Ennemose. "3D-CFD diesel combustion and accurate heat transfer modeling for diesel engines." *Proceedings of THIESEL* 2002 (2002): 445-454.
[38] Suzzi, Daniele, Eberhard von Berg, J. V. Pastor, Gian Marco Bianchi, and Reinhard Tatschl. "Simulation of Primary Break-up of Diesel Jets by a Hybrid Method combining VOF-calculations and the Classical DDM Rate Approach with a 3D CFD Code." In *Proceeding ILASS 2004 Conference*. 2004.
[39] Shi, Yu, and Rolf D. Reitz. "Optimization study of the effects of bowl geometry, spray targeting, and swirl ratio for a heavy-duty diesel engine operated at low and high load." *International Journal of Engine Research* 9, no. 4 (2008): 325-346. https://doi.org/10.1243/14680874JER00808
[40] Park, Sungwook. "Optimization of combustion chamber geometry and engine operating conditions for compression ignition engines fueled with dimethyl ether." *Fuel* 97 (2012): 61-71. https://doi.org/10.1016/j.fuel.2012.03.004
[41] Sweetland, P., and Rolf D. Reitz. *Particle image velocimetry measurements in the piston bowl of a DI diesel engine*. No. 940283. SAE Technical Paper, 1994. https://doi.org/10.4271/940283
[42] Erdil, A., and A. Kodal. "A comparative study of turbulent velocity fields in an internal combustion engine with shrouded valve and flat/bowl piston configurations." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 221, no. 12 (2007): 1597-1607. https://doi.org/10.1243/09544062JMES762
[43] Maji, Luo, Xiang Liangshan, and Li Xiong. "Numerical simulation of the effects of combustion chamber geometry on nonroad diesel engine performance." In *2011 International Conference on Electric Information and Control Engineering*, pp. 2407-2411. IEEE, 2011.
[44] Rakopoulos, C. D., G. M. Kosmadakis, and 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, no. 3 (2010): 470-484. https://doi.org/10.1016/j.enconman.2009.10.010

[45] Chui, E. H., P. MJ Hughes, and G. D. Raithby. "Implementation of the finite volume method for calculating radiative transfer in a pulverized fuel flame." *Combustion Science and Technology* 92, no. 4-6 (1993): 225-242. https://doi.org/10.1080/00102209308907673
[46] Abdul Rahiman, Abdul Razak R. K., A. D. Mohammad Samee, and M. K. Ramis. "CFD analysis of flow field development in a direct injection diesel engine with different manifolds." *American Journal of Fluid Dynamics* 4, no. 3 (2014): 102-113.
[47] Toh, Hoong T., Rong F. Huang, Kuan H. Lin, and Ming-Jyh Chern. "Computational study on the effect of inlet port configuration on in-cylinder flow of a motored four-valve internal combustion engine." *Journal of Energy Engineering* 137, no. 4 (2011): 198-206. https://doi.org/10.1061/(ASCE)EY.1943-7897.0000044
[48] Deshmukh, Devendra, Rajesh Kumar, Manish Garg, Mohammad Jaffer Nayeem, and V. Lakshminarasimhan. "Optimisation of gas exchange process on a single cylinder small 4-stroke engine by intake and exhaust tuning: Experimentation and simulation." *SAE Transactions* (2004): 1741-1749. https://doi.org/10.4271/2004-32-0007
[49] Sammut, Gilbert, and Alex C. Alkidas. *Relative contributions of intake and exhaust tuning on SI engine breathing-A computational study*. No. 2007-01-0492. SAE Technical Paper, 2007. https://doi.org/10.4271/2007-01-0492
[50] Vítek, Oldřich, and Miloš Polášek. "Tuned Manifold Systems-Application of 1-D Pipe Model." *SAE Transactions* (2002): 271-280. https://doi.org/10.4271/2002-01-0004
[51] Taylor, James, David Gurney, Paul Freeland, Rene Dingelstadt, Jürgen Stehlig, and Veit Bruggesser. *Intake manifold length effects on turbocharged gasoline downsizing engine performance and fuel economy*. No. 2012-01-0714. SAE Technical Paper, 2012. https://doi.org/10.4271/2012-01-0714
[52] Samuel, Jensen, N. S. Prasad, and Kumarasamy Annamalai. *Effect of Variable Length Intake Manifold on a Turbocharged Multi-Cylinder Diesel Engine*. No. 2013-01-2756. SAE Technical Paper, 2013. https://doi.org/10.4271/2013-01-2756
[53] Bari, Saiful, and Idris Saad. "CFD modelling of the effect of guide vane swirl and tumble device to generate better in-cylinder air flow in a CI engine fuelled by biodiesel." *Computers & Fluids* 84 (2013): 262-269. https://doi.org/10.1016/j.compfluid.2013.06.011
[54] Ramajo, Damian E., and Norberto M. Nigro. "In-Cylinder Flow Computational Fluid Dynamics Analysis of a Four-Valve Spark Ignition Engine: Comparison Between Steady and Dynamic Tests." *Journal of Engineering for Gas Turbines and Power* 132, no. 5 (2010): 052804. https://doi.org/10.1115/1.4000265
[55] Liu, Kai, and D. C. Haworth. "Large-eddy simulation for an axisymmetric piston-cylinder assembly with and without swirl." *Flow, Turbulence and Combustion* 85 (2010): 279-307. https://doi.org/10.1007/s10494-010-9292-1
[56] Aziz, A. Rashid A., and Raja Shahzad. "Combustion analysis of a CNG direct injection spark ignition engine." *International Journal of Automotive and Mechanical Engineering* 2 (2010): 157-170. https://doi.org/10.15282/ijame.2.2010.5.0013
[57] Hamada, Khalaf I., M. M. Rahman, M. A. Abdullah, Rosli A. Bakar, and A. Rashid A. Aziz. "Effect of mixture strength and injection timing on combustion characteristics of a direct injection hydrogen-fueled engine." *International Journal of Hydrogen Energy* 38, no. 9 (2013): 3793-3801. https://doi.org/10.1016/j.ijhydene.2013.01.092
[58] Pundir, B. P. *Engine emissions: pollutant formation and advances in control technology*. Alpha Science International, Limited, 2007.
[59] Rahman, S. M. Ashrafur, H. H. Masjuki, M. A. Kalam, A. Sanjid, and M. J. Abedin. "Assessment of emission and performance of compression ignition engine with varying injection timing." *Renewable and Sustainable Energy Reviews* 35 (2014): 221-230. https://doi.org/10.1016/j.rser.2014.03.049
[60] Hwang, Joonsik, Donghui Qi, Yongjin Jung, and Choongsik Bae. "Effect of injection parameters on the combustion and emission characteristics in a common-rail direct injection diesel engine fueled with waste cooking oil biodiesel." *Renewable Energy* 63 (2014): 9-17. https://doi.org/10.1016/j.renene.2013.08.051
[61] Liu, Junheng, Anren Yao, and Chunde Yao. "Effects of injection timing on performance and emissions of a HD diesel engine with DMCC." *Fuel* 134 (2014): 107-113. https://doi.org/10.1016/j.fuel.2014.05.075
[62] Wang, Ying, Yuwei Zhao, Fan Xiao, and Dongchang Li. "Combustion and emission characteristics of a diesel engine with DME as port premixing fuel under different injection timing." *Energy Conversion and Management* 77 (2014): 52-60. https://doi.org/10.1016/j.enconman.2013.09.011
[63] Agarwal, Avinash Kumar, Dhananjay Kumar Srivastava, Atul Dhar, Rakesh Kumar Maurya, Pravesh Chandra Shukla, and Akhilendra Pratap Singh. "Effect of fuel injection timing and pressure on combustion, emissions and performance characteristics of a single cylinder diesel engine." *Fuel* 111 (2013): 374-383. https://doi.org/10.1016/j.fuel.2013.03.016
[64] Liu, Kai, Daniel C. Haworth, Xiaofeng Yang, and Venkatesh Gopalakrishnan. "Large-eddy simulation of motored flow in a two-valve piston engine: POD analysis and cycle-to-cycle variations." *Flow, Turbulence and Combustion* 91 (2013): 373-403. https://doi.org/10.1007/s10494-013-9475-7

[65] Salvador, F. J., Jorge Martínez-López, M. Caballer, and C. De Alfonso. "Study of the influence of the needle lift on the internal flow and cavitation phenomenon in diesel injector nozzles by CFD using RANS methods." *Energy Conversion and Management* 66 (2013): 246-256. https://doi.org/10.1016/j.enconman.2012.10.011

[66] Yang, Jie, Xue Dong, Qiang Wu, and Min Xu. "Effects of enhanced tumble ratios on the in-cylinder performance of a gasoline direct injection optical engine." *Applied Energy* 236 (2019): 137-146. https://doi.org/10.1016/j.apenergy.2018.11.059

[67] Pathak, Bhavesh, and Nikul Patel. "Thermodynamic Performance of an Engine by Modifying Piston Bowl Geometries Fuelled by SME-100, LA-100, KB-100 Biodiesel Blends, and Diesel." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 102, no. 1 (2023): 1-13. https://doi.org/10.37934/arfmts.102.1.113

[68] Harshavardhan, Ballapu, and J. M. Mallikarjuna. "Effect of piston shape on in-cylinder flows and air-fuel interaction in a direct injection spark ignition engine-A CFD analysis." *Energy* 81 (2015): 361-372. https://doi.org/10.1016/j.energy.2014.12.049

[69] Nicollet, Franck, Christian Krüger, Jürgen Schorr, Edouard Nicoud, Olivier Colin, Christian Angelberger, Johannes Bode, and Benjamin Böhm. "A PIV-guided large-eddy simulation of in-cylinder flows." *Oil & Gas Science and Technology-Revue d'IFP Energies nouvelles* 72, no. 5 (2017): 28. https://doi.org/10.2516/ogst/2017022

[70] Jaichandar, S., and K. Annamalai. "Influences of re-entrant combustion chamber geometry on the performance of Pongamia biodiesel in a DI diesel engine." *Energy* 44, no. 1 (2012): 633-640. https://doi.org/10.1016/j.energy.2012.05.029

[71] Jaichandar, S., and K. Annamalai. "Effects of open combustion chamber geometries on the performance of pongamia biodiesel in a DI diesel engine." *Fuel* 98 (2012): 272-279. https://doi.org/10.1016/j.fuel.2012.04.004

[72] Raj, Antony Raj Gnana Sagaya, Jawali Maharudrappa Mallikarjuna, and Venkitachalam Ganesan. "Energy efficient piston configuration for effective air motion-A CFD study." *Applied Energy* 102 (2013): 347-354. https://doi.org/10.1016/j.apenergy.2012.07.022

[73] Mobasheri, Raouf, and Zhijun Peng. "CFD investigation of the effects of re-entrant combustion chamber geometry in a HSDI diesel engine." *World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering* 7, no. 4 (2013): 770-780.

[74] Kidoguchi, Yoshiyuki, Changlin Yang, and Kei Miwa. *Effect of high squish combustion chamber on simultaneous reduction of NOx and particulate from a direct-injection diesel engine*. No. 1999-01-1502. SAE Technical Paper, 1999. https://doi.org/10.4271/1999-01-1502

[75] Subramanian, Saravanan, Balamurugan Rathinam, J. Isaac Joshua Ramesh Lalvani, and Kandaswamy Annamalai. *Piston bowl optimization for single cylinder diesel engine using CFD*. No. 2016-28-0107. SAE Technical Paper, 2016. https://doi.org/10.4271/2016-28-0107

[76] Gafoor, CP Abdul, and Rajesh Gupta. "Numerical investigation of piston bowl geometry and swirl ratio on emission from diesel engines." *Energy Conversion and Management* 101 (2015): 541-551. https://doi.org/10.1016/j.enconman.2015.06.007

[77] Prasad, B. V. V. S. U., C. S. Sharma, T. N. C. Anand, and R. V. Ravikrishna. "High swirl-inducing piston bowls in small diesel engines for emission reduction." *Applied Energy* 88, no. 7 (2011): 2355-2367. https://doi.org/10.1016/j.apenergy.2010.12.068

[78] da Costa, Roberto Berlini Rodrigues, Raphael Meireles Braga, Carlos Alberto Gomes Júnior, Ramón Molina Valle, and Rudolf Huebner. "PIV measurements and numerical analysis of in-cylinder tumble flow in a motored engine." *Journal of the Brazilian Society of Mechanical Sciences and Engineering* 39 (2017): 3931-3945. https://doi.org/10.1007/s40430-017-0878-6

[79] Azad, A. K., Pobitra Halder, K. Nanthagopal, and B. Ashok. "Investigation of diesel engine in cylinder flow phenomena using CFD cold flow simulation." In *Advanced Biofuels*, pp. 329-336. Woodhead Publishing, 2019. https://doi.org/10.1016/B978-0-08-102791-2.00013-1

[80] Krishna, Addepalli S., J. M. Mallikarjuna, and Davinder Kumar. "Effect of engine parameters on in-cylinder flows in a two-stroke gasoline direct injection engine." *Applied Energy* 176 (2016): 282-294. https://doi.org/10.1016/j.apenergy.2016.05.067

[81] Zhang, Xianhui, Tianyou Wang, Ming Jia, Wei Li, Lei Cui, and Xigang Zhang. "The interactions of in-cylinder flow and fuel spray in a gasoline direct injection engine with variable tumble." *Journal of Engineering for Gas Turbines and Power* 137, no. 7 (2015): 071507. https://doi.org/10.1115/1.4029208

[82] Chen, Hao, David L. Reuss, David LS Hung, and Volker Sick. "A practical guide for using proper orthogonal decomposition in engine research." *International Journal of Engine Research* 14, no. 4 (2013): 307-319. https://doi.org/10.1177/1468087412455748

[83] Nishad, Kaushal, Florian Ries, Yongxiang Li, and Amsini Sadiki. "Numerical investigation of flow through a valve during charge intake in a DISI-engine using large eddy simulation." *Energies* 12, no. 13 (2019): 2620. https://doi.org/10.3390/en12132620

[84] Keskinen, Jukka-Pekka, Ville Vuorinen, Ossi Kaario, and Martti Larmi. "Large eddy simulation of a piston-cylinder assembly: The sensitivity of the in-cylinder flow field for residual intake and in-cylinder velocity structures." *Computers & Fluids* 122 (2015): 123-135. https://doi.org/10.1016/j.compfluid.2015.08.028

[85] Hartmann, Frank, Stefan Buhl, Florian Gleiss, Philipp Barth, Martin Schild, Sebastian A. Kaiser, and Christian Hasse. "Spatially resolved experimental and numerical investigation of the flow through the intake port of an internal combustion engine." *Oil & Gas Science and Technology-Revue d'IFP Energies nouvelles* 71, no. 1 (2016): 2. https://doi.org/10.2516/ogst/2015022

[86] Buhl, Stefan, Frank Hartmann, Sebastian A. Kaiser, and Christian Hasse. "Investigation of an IC engine intake flow based on highly resolved LES and PIV." *Oil & Gas Science and Technology-Revue d'IFP Energies nouvelles* 72, no. 3 (2017): 15. https://doi.org/10.2516/ogst/2017012

[87] di Mare, Francesca, Robert Knappstein, and Michael Baumann. "Application of LES-quality criteria to internal combustion engine flows." *Computers & Fluids* 89 (2014): 200-213. https://doi.org/10.1016/j.compfluid.2013.11.003

[88] Baum, E., B. Peterson, B. Böhm, and A. Dreizler. "On the validation of LES applied to internal combustion engine flows: part 1: comprehensive experimental database." *Flow, Turbulence and Combustion* 92 (2014): 269-297. https://doi.org/10.1007/s10494-013-9468-6

[89] Baumann, M., F. Di Mare, and J. Janicka. "On the validation of large eddy simulation applied to internal combustion engine flows part II: numerical analysis." *Flow, Turbulence and Combustion* 92 (2014): 299-317. https://doi.org/10.1007/s10494-013-9472-x

[90] Anand, M. S., R. Eggels, M. Staufer, M. Zedda, and J. Zhu. "An advanced unstructured-grid finite-volume design system for gas turbine combustion analysis." In *Gas Turbine India Conference*, vol. 35161, p. V001T03A003. American Society of Mechanical Engineers, 2013. https://doi.org/10.1115/GTINDIA2013-3537