

Effect of Engine Parameters on Mixture Stratification in a Wall-Guided GDI Engine - A Quantitative CFD Analysis

Krishna S Addepalli and J M Mallikarjuna
 Indian Institute of Technology Madras

ABSTRACT

Today, GDI engines are becoming very popular because of better fuel economy and low exhaust emissions. The gain in fuel economy in these engines is realized only in the stratified mode of operation. In wall-guided GDI engines, the mixture stratification is realized by properly shaping the combustion chamber. However, the level of mixture stratification varies significantly with engine operating conditions. In this study, an attempt has been made to understand the effect of engine operating parameters viz., compression ratio, engine speed and inlet air pressure on the level of mixture stratification in a four-stroke wall-guided GDI engine using CFD analysis. Three compression ratios of 10.5, 11.5 and 12.5, three engine speeds of 2000, 3000 and 4000 rev/min., and three inlet air pressures of 1, 1.2 and 1.4 bar are considered for the analysis. The CONVERGE software is used to perform the CFD analysis. Simulation is done for one full cycle of the engine. The CFD models used are validated with the available experimental results in the literature to the extent possible. The mixture stratification is quantitatively estimated by a new method developed by the authors. From the results, it is found that, the mixture stratification is better at low engine speeds and low inlet air pressure. Also, it is found that higher stratification index (SI) resulted in low heat release rate.

KEYWORDS: GDI engine, mixture distribution, engine parameters, fuel stratification

CITATION: Addepalli, K. and Mallikarjuna, J., "Effect of Engine Parameters on Mixture Stratification in a Wall-Guided GDI Engine - A Quantitative CFD Analysis," *SAE Int. J. Commer. Veh.* 10(2):2017, doi:10.4271/2017-01-0570.

1. INTRODUCTION

Today, spark ignition and compression ignition engines; dominate the powertrain segment of automotive companies [1]. While generating power by burning fuel, both spark and compression ignition engines generate various types of harmful emissions as by-products. The rapid depletion of fossil fuels and the concern of environmental pollution, led engine researchers to focus on improving the fuel economy and reducing exhaust emissions [2, 3, 4, 5, 6, 7]. In this direction, one of the recent options is the GDI technology in spark ignited engines [8, 9, 10]. In GDI engines, gasoline is directly injected into the combustion chamber either during the suction stroke or the compression stroke, depending on the power output. Injecting the fuel, in the suction stroke, creates a mixture which is almost homogenous, at the time of spark discharge, owing to the longer time available for mixing. When the fuel is injected into the engine cylinder during the compression stroke, it does not have sufficient time to mix to with the air to form homogenous mixture [11]. Thus, the late fuel injection forms a stratified mixture in the cylinder, where fuel vapor is carried towards the spark plug location such that a slightly rich mixture exists there; with a very lean mixture at other locations and an over lean mixture in the cylinder [12]. Generally, the fuel vapors are carried towards the spark plug location from the fuel injector in three ways. Based on this, the GDI engines are categorized as spray guided, air guided and wall guided types. In the wall guided type, the fuel is sprayed on to a specially designed piston surface, which will carry it towards the spark

plug location at the time of spark. Hence, the piston top surface in this engine has to be designed in such a way that, it deflects the fuel towards the spark plug location over a range of engine operating conditions [12]. The physical and chemical properties of the fuel, engine geometric and operating parameters play a vital role in deciding the level of mixture stratification.

Therefore understanding the mixture stratification in GDI engines is very vital and has good scope. Previously, *Song et al.*, [13] studied the effect of single and split injection strategies on mixture preparation and combustion characteristics of a DISI engine. The single fuel injection strategy was realized with a wide range of fuel injection timing during the intake stroke and the double fuel injection was realized with short second fuel injection, during the compression stroke. Numerical simulations were carried out using KIVA to observe the in-cylinder flow and fuel spray spreading procedure. The flame images were obtained by a high-speed CMOS camera and in-cylinder pressure traces were used to evaluate the combustion characteristics. They found that the peak in-cylinder pressure, turbulence intensity and flame radius increased with retarded fuel injection timings. They also found that stronger turbulence, at the time of spark, led to faster combustion and higher in-cylinder pressure. They also observed that the flame propagation direction was mainly influenced by the ER distribution and the flame always propagated towards fuel rich zones.

Yin et al., [14] studied the effect of piston top contour on tumble flow, mixture distribution and combustion characteristics in a GDI engine equipped with CMCV using CFD. The four piston shapes used in this study were flat, bowl, flat with bowl in the center and dual offset bowls. They found that the bowl piston enhanced the tumble flow formation and development, and hence gave maximum combustion efficiency when used along with CMCV. They also found that the flat piston performed well at lower engine speeds, while the performance deteriorated at higher engine speeds. They analyzed the mixture homogeneity using a quantity called 'uniformity index'. They concluded that the closed CMCV contributed to faster fuel evaporation rate at lower engine speeds.

Keskinen et al., [15] studied the effect of nozzle type, fuel injection pressure and timing on the mixture formation in a direct injection gas engine using CFD. They found that the fuel-air mixing rate was more influenced by the fuel injection timing than the fuel injection pressure. They identified two phases of fuel spray evolution in the engine cylinder; the first phase was characterized by a rapid fuel spread in the cylinder and the second phase was characterized by the mixture homogenization. They found that the most homogeneous mixture in the engine cylinder was achieved with a hollow cone nozzle with early injection or a circular nozzle with late injection.

Costa et al., [16] studied the mixture formation and early flame development in a wall-guided, research GDI engine using both numerical technique and UV-digital imaging. They found that the flame propagation and combustion stability mainly depended on the mixture formation, which in turn depended on the shape of the combustion chamber. They concluded that the early fuel injection which led to the homogenous mixture in the combustion chamber at the time of spark discharge resulted in a stable combustion. They also found that the late fuel injection to achieve stratified mixture led to high cycle-by-cycle variations.

Giovannoni et al., [17] studied the effect of fuel composition and injection strategy on the mixture preparation and soot formation in a GDI engine using CFD. They compared the tailpipe soot emissions for a conventional single-component fuel approach and a more refined multi-component fuel approach. The multi-component fuel blend was modeled using a discrete-continuous multi-component approach. They also compared the mixture distribution for the various SOI timings on a vertical plane passing through the spark plug. They concluded that a correct calibration of droplet/wall interaction model was compulsory to obtain accurate liquid film deposits and soot emissions. They also concluded that the multi-component fuel modeling approach gave more accurate soot predictions.

Martins and De Souza [18] analyzed the air flow characterization, mixture formation and stratification in a DISI engine using CFD. New bowl geometry was proposed for the engine considered to obtain the deflection of injected fuel towards the spark plug by the gas jet for maintaining the stoichiometric or rich equivalence ratio in

that region. They also studied the effect of swirl ratio on the distribution of turbulent kinetic energy and charge stratification in the engine cylinder.

Liu et al., [19] developed a new multi-dimensional numerical simulation method to analyze the air-fuel mixing, stratified combustion and NOx emission formation in a GDI engine. Many commonly used CFD sub-models for GDI engine simulations were integrated into the developed simulation method. After calibrating the developed simulation method with the experimental data, they studied the effect of piston bowl offset on the mixture preparation, engine performance, and NOx emissions. They recommended a piston bowl offset of 0.5 to 1 cm, to obtain an optimum fuel burning rate and NOx emissions.

Many approaches were used by the previous researchers in order to study the mixture stratification in a GDI engine with both optical diagnostics and the numerical techniques. But, most of them are either qualitative or insufficient to understand and predict their effect on combustion and emissions. In this study, the mixture stratification is quantitatively expressed with the help of a new method developed by the authors. Then the effect of different engine operating parameters like engine speed, compression ratio and the intake air pressure on the mixture stratification is analyzed.

2. GEOMETRIC MODELLING

In this study, a four-stroke wall guided GDI engine is considered for the CFD analysis. The specifications of the engine considered are shown in Table 1. These engine specifications are taken from Costa et al., [16]. The engine geometry used for the CFD analysis is shown in Figure 1. In this study, except for initial validation of the CFD results, a pent roof with an offset bowl piston as shown in Figure 2 is used for the CFD analysis, as opposed to the bow-ditch piston used by Costa et al., [16] to enhance the performance at the stratified mode of operation.

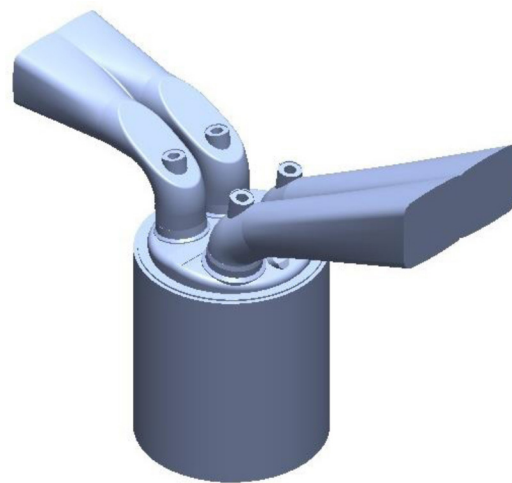


Figure 1. Geometrical model of the engine considered

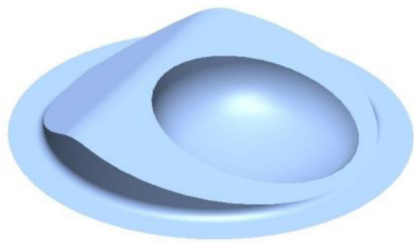


Figure 2. A view of the top surface of the piston used

Table.1. Engine specifications

Stroke	79 mm
Bore	81.3 mm
Connecting Rod	143 mm
Compression ratio	10.6 : 1
Number of Valves	4
Exhaust Valve Open (EVO)	27° before BDC
Exhaust Valve Close (EVC)	0° after TDC
Inlet Valve Open (IVO)	3° before TDC
Inlet Valve Close (IVC)	36° after BDC
Fuel system	Direct injection, 100 bar

3. CFD ANALYSIS

In this study, the CFD analysis is carried out using commercial CFD software, CONVERGE. In this, the grid generation of the engine geometry is performed during the run time using the cut-cell Cartesian meshing method. Here, the base mesh size is kept at 1 mm. The spatial mesh refinement, at the locations, like the injector and spark plug, is performed using the fixed embedding technique for a specified period of crank angle. Apart from this, the adaptive grid refinement technique is used to capture the velocity and temperature gradients as and when they cross a particular threshold. The overall grid size varies between 0.125 to 1 mm in the entire computational domain at any given time of the simulation.

The flow turbulence is analyzed using RNG k- ϵ model [20-21]. Fuel spray breakup and atomization is analyzed using kh-rt model [22]. Combustion analysis is performed using SAGE, which is a detailed chemistry solver. Spray turbulent dispersion is modeled using the O'Rourke model and the droplet evaporation is analyzed using the Frossling model [23]. All the simulations are carried out for one full cycle of 720 CAD starting from EVO. The time step varies between 10^{-8} and 10^{-5} seconds depending on the grid CFL criteria. The pressure-velocity coupling is carried out using PISO algorithm that is applied to a collocated grid using the Rhie-Chow scheme [24]. The NTC collision model is used to analyze droplet collisions [25] and the droplet wall interaction is analyzed using the O'Rourke wall collision model [26]. All the CFD simulations are carried out at the stratified condition with an overall ER of 0.75 ± 0.05 . The engine considered in this study basically operates at a compression ratio of 10.5, engine speed of 2000 rev/min and an intake air pressure of 1 bar. Whenever one of the above mentioned conditions are varied to study its effect on mixture stratification, the other conditions are kept constant. The CFD mesh on the central vertical plane at the instant of spark (700 CAD) is shown in Figure 3.

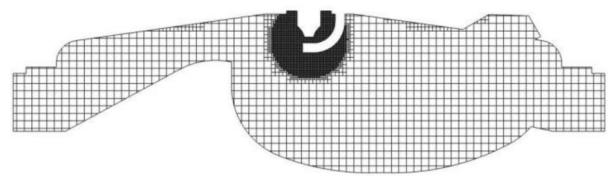


Figure 3. Meshing on the central vertical plane at 700 CAD

4. VALIDATION OF CFD MODELS

In this study, the CFD models used are validated with the available experimental results from the literature, for the same engine configuration [16]. The variation of pressure with the crank angles, the ER distribution on the central vertical plane at 710 CAD and the variation of the mass fraction of carbon dioxide with crank angles, obtained by Costa et al., [16] are compared with those of the present CFD study.

Figure 4 shows the comparison of the variation of in-cylinder pressure with crank angles among the experimental results of Costa et al., [16] and the present CFD study. The experimental pressure trace is an average of 200 consecutive cycles whereas the CFD simulation results are the obtained from the fourth cycle of a multiple cycle simulation. From Figure 4, it is seen that the two results are in good agreement. Therefore, the CFD models of this study can be used for further analysis with confidence.

Figure 5 shows the comparison of the spatial distribution of ER at the time of spark (10 CAD before TDC) between the simulation results of Costa et al., [16] and the present CFD study when the fuel injection timing is 300 CAD before TDC. From Figure 5, it is seen that, the ER distribution obtained by the present CFD simulations is very much similar to that obtained by Costa et al. [16].

Figure 6 shows the comparison of the mass fraction of carbon dioxide between the simulation results of Costa et al., [16] and the present CFD study. From Figure 6, it is seen that, the variation of mass fraction of carbon dioxide with crank angles, obtained by the present CFD simulations is very similar to that obtained by Costa et al. [16], both qualitatively and quantitatively.

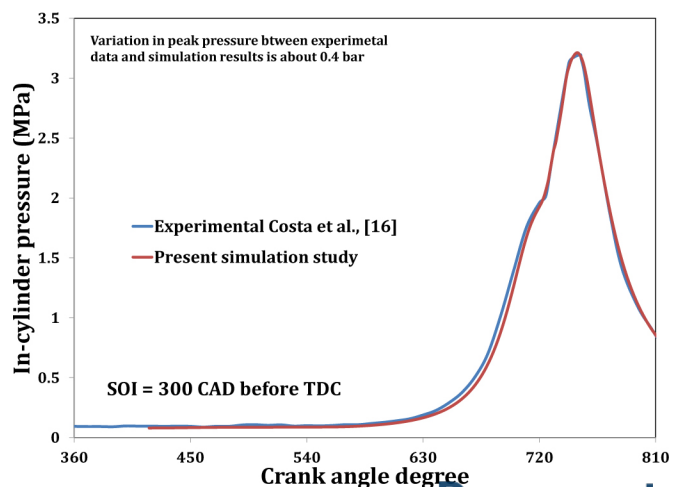


Figure 4. Comparison of in-cylinder pressure with crank angle for SOI of 300 CAD before TDC