Scientia Iranica B (2013) 20(6), 1729–1741

SCIENTIA IRANICA

Sharif University of Technology
Scientia Iranica
Transactions B: Mechanical Engineering
www.scientiairanica.com

Multidimensional modeling of CNG direct injection and mixture preparation in a SI engine cylinder

B. Yadollahi and M. Boroomand*

Department of Aerospace Engineering, Amirkabir University of Technology, Tehran, P.O. Box 15875-4413, Iran.

Received 16 March 2012; received in revised form 10 December 2012; accepted 22 July 2013

KEYWORDS
CNG direct injection;
Spark ignition;
Mixture preparation;
DISI;
High pressure injection;
Cylinder head shape;
Multidimensional modeling;
Numerical simulation.

Abstract. In this study, a numerical model has been developed with AVL FIRE software to perform an investigation into a direct natural gas injection into the cylinder of spark ignition internal combustion engines. Two main parts have been taken into consideration, aiming to convert an MPFI gasoline engine to a direct injection NG engine. In the first part, multi-dimensional numerical simulation of the transient injection process, the mixing and the flow field have been performed using different validation cases in order to assure the numerical model validity of results. In all cases, the present results were found to have excellent agreement with experimental and numerical results from literature. In the second part, the validated model has been applied to methane injection into the cylinder of a direct injection engine. Five different piston head shapes have been taken into consideration and a centrally mounted inwardly-opening single-hole injector has been adapted to all cases. The effects of injection and combustion chamber geometry have been studied on the mixing of air-fuel inside the cylinder via quantitative and qualitative representation of results. Furthermore, the effect of real engine cylinder head shape on gas injection has been investigated. Based on the results, suitable geometrical configuration for a NG DI engine has been discussed.

1. Introduction

In general, the flow field inside a cylinder is quite complex. The direct injection of fuel makes it more complex, and the situation turns even more complicated with gaseous fuels. Due to its low density, mixture formation with natural gas is more critical in comparison to liquid fuel engines. The reason is that even very high injection velocities produce low fuel penetration and, consequently, poor mixture formation. Thus, mixture formation in gas engines depends more on in-cylinder charge motion than in gasoline engines [1].

To enhance NG fuel penetration in a cylinder, very high fuel rail pressures of up to 200 bar are used. Such an elevated pressure can also help increase the turbulence level of the mixture and the overall fuel-air mixing.

Very high pressure ratios between the fuel rail and the in-cylinder cause the flow at the injector exit to be typically under-expanded. This means that the flow becomes sonic at the nozzle exit and undergoes a complex pattern of shock-expansion waves downstream from the nozzle exit.

Accurate capturing of such a pattern in the near field has been shown to strongly affect the complete jet shape and mixture formation downstream from the nozzle [2]. Huang et al. [3-6] conducted a set of experimental studies on the different injection and combustion characteristics of natural gas using a rapid compression machine and also a natural gas fuelled engine.

*. *Corresponding author. Tel: +98 21 64543280; Fax: +98 21 66959020*
E-mail addresses: Byadollahi@aut.ac.ir (B. Yadollahi), Boromand@aut.ac.ir (M. Boroomand)

Numerical investigation of the flow in these engines has been focused on different challenges of modeling. Multi-dimensional modeling, with the focus on turbulence modeling, has been performed in [7-9], and evolution of gas jet injection is studied in [7,10,11].

Li et al. [2] conducted a 2D numerical modeling, as well as method of characteristics, to study under-expanded gas injection into the cylinder of a large bore engine. In the study, it was reported that the main parameters determining the type of flow exiting a sonic nozzle are the pressure expansion ratio and the nozzle geometry. It has been mentioned that accurate computation of the flow at the nozzle exit is of main importance because it controls the jet mixing downstream from the nozzle.

In the same study, and also in [11,12], grid density at the nozzle exit has been discussed. It has been reported that a minimum of about 10 cell layers is necessary for accurate capturing of a flow field in the near field.

Ouellette and Hill [11] studied turbulent transient methane and air injection into a constant volume chamber. A multi-dimensional numerical, as well as experimental, investigation was performed on subsonic and sonic jets emerging from a single nozzle into the chamber. The numerical modeling tool used in the study was the KIVA code, and jet shape and penetration were presented in different cases.

A virtual nozzle approach has been used to model the injection instead of computing the detailed near field flow in [10,13-15]. Such cases have their inlet boundary condition set at the mach disc in the nozzle exit, and, thus, the inside injector space is eliminated. This method results in decreased grid density at the nozzle exit but is limited to two dimensional slot and hole injector geometries. It also requires accurate data regarding mach disc size and location.

The idea of fictive droplets of gas was introduced into a numerical model using Quicksim software to reduce the grid number requirements of the modeling (see [1]). Also, in [16], a phenomenological model has been implemented along with the KIVA code to predict the quantities of the flow in the near field.

It could be concluded from the literature that the most accurate approach is to include inside-injector space into computations. Such an approach is followed by [17,18]. In [17], a multi-dimensional modeling approach was undertaken using STAR-CD software. A centrally mounted outwardly-opening injector was implemented in the model, and injection and mixing were studied in different poppet-valve chamber geometries. The injection process has also been studied in a type of single-cylinder research engine.

In the authors' previous publications [19,20], the evolution of gas jet into the cylinder of a direct injection engine was studied using AVL FIRE software. The effects of injection, piston head geometry, injector type and engine speed have been investigated in the studies.

Table 1. Base engine characteristics.

Bore	83 mm
Stroke	81.4 mm
Displacement volume	440 cc/cyl
Connecting rod length	150.15 mm
Compression ratio	9.5

2. Present study

It is intended to convert an existing 1800 cc MPFI gasoline engine to a direct injection NG engine with minimum modifications. In the first phase of the project, multi-dimensional numerical simulation of the transient injection process, the mixing and the flow field has been performed. Table 1 illustrates the basic characteristics of the engine. A view of the base engine geometry is shown in Figure 1.

There are different aspects of flow inside a cylinder which should be studied. Two of the most important parts of these aspects are combustion chamber geometry and injection studies, which have been taken into consideration in the present work. The objective is to determine a suitable configuration for the piston head geometry, with regard to injector type and location.

To investigate the effects of injector and combustion chamber geometry, a 3D numerical model has been developed in AVL FIRE. The study is focused on injector and combustion chamber mixing capabilities in DI NG engines. To perform modeling, a centrally mounted, inwardly-opening single-hole injector has been chosen.

The mixture preparation for DI engines lies in two main strategies. First, a homogenous charge mixture preparation with early injection in the intake stroke, and second, a stratified charge mixture preparation with late injection in the compression stroke. The latter is reportedly a more efficient strategy in terms of fuel economy [1,17], so, it is intended to use a stratified charge mixture preparation strategy in the modifications.

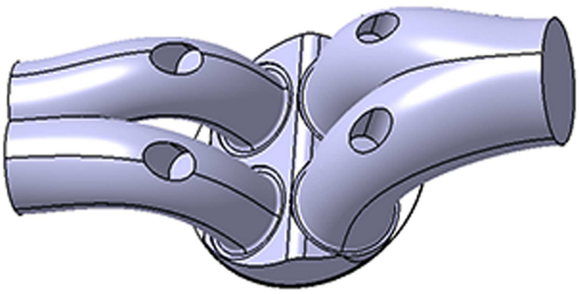


Figure 1. A view of base engine geometry and ports.

The study has been conducted through two main parts. Firstly, a numerical model has been developed to evaluate the software capability in natural gas direct injection modeling. To achieve this, validation case studies have been undertaken to make sure the numerical model can predict the injection and flow physics adequately. The studies in this part mainly include transient turbulent gas injection in a constant volume chamber. In each case, the calculation results have been compared with experimental data to validate the numerical modeling.

Secondly, flow inside a real engine cylinder has been studied, aiming to get a flammable stratified mixture. In this part, different combustion chamber geometries have been considered, and injection and mixture formations in each case have been studied. To comply with real engine conditions, i.e. variable volume and moving boundaries, in this part, a moving grid technique should be adopted. In each case, different representations of the results have been made and based on the results, discussion on the suitability of each model has been conducted.

To determine the mixing performance of any injector-geometry configuration, as mentioned in [17], the flammability of the mixture in any computational finite volume cell has been categorized in three different levels of: too lean, flammable and too rich. A flammable mass fraction has been calculated for each cell on this basis. The values of 0.8 and 1.4 for RAFR have been chosen for flammability limits, as suggested in the same study.

Finally, to determine the effects of real engine cylinder head shape on the mixture preparation, a set of investigations were performed.

3. Numerical model development

Although the most accurate way to model the flow in the near field of injector nozzles is to take inside-injector space into account, detailed numerical calculation of the inside nozzle flow requires significant computational effort. The reason lies within the difference between the cylinder and nozzle length scales, which could be in the order of 1000.

Generally, the natural gas injector holes have very small diameters of, typically, less than 1 mm. One of the most important variables affecting the numerical models ability to predict the flow correctly is the number of grid points across such a small nozzle diameter. To investigate the effect of the number of grid points across the nozzle diameter, the study by Chiodi et al. [1] has been taken into consideration. In this case, a couple of numerical models with different numbers of grid points across the nozzle diameter have been generated. The modeling conditions are shown in Table 2.

Table 2. Injection conditions [1].

Injected fluid	Methane
Chamber fluid	Air
Fuel rail pressure	8 bar
Chamber pressure	1 bar
Chamber temperature	300 k
Duration of injection	10 ms
Fuel mass flow	270 mg/s

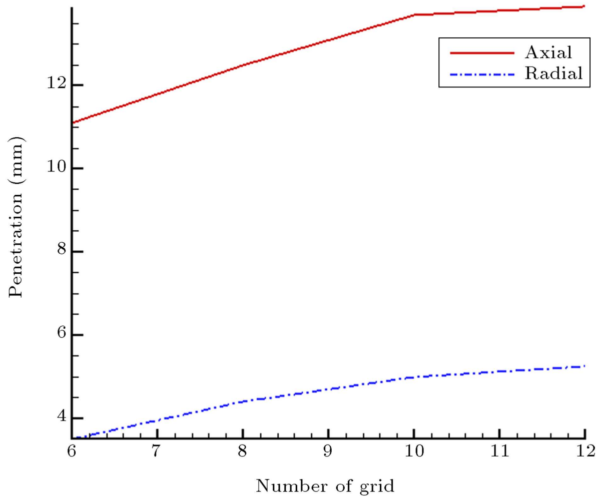


Figure 2. Variations of axial and radial penetration using different number of grid points across the nozzle.

The results for the penetration of the jet into the chamber for time 0.1 ms ASI have been plotted against the number of grid points across the nozzle in Figure 2. The reason to check penetration in the first time step is realized regarding the effect of the number of grid points across the nozzle diameter on flow in the near field of the nozzle.

It can be seen from the figure that penetration increases with an increase in the number of grid points. But, increasing the number of grid points across the nozzle to more than 10 layers seems to have little effect on penetration. This suggests that 10 cell layers across the nozzle are appropriate for grid-independent numerical simulation of CNG injection. This result agrees with the results reported by [2]. This number of grid points has been used for all cases of injection in the present study.

In order to better investigate the effect of turbulent mixing inside the cylinder, the velocity distribution across the jet at periods of 0.2 ms and 0.8 ms ASI has been depicted in Figure 3. It can be seen in the figure that increasing the cell layers to more than 10 layers does not have a significant effect on the prediction of flow velocity on two consecutive planes at 2 mm and 5 mm, downstream of the nozzle exit. This verifies the fact that in order to achieve grid independency, a

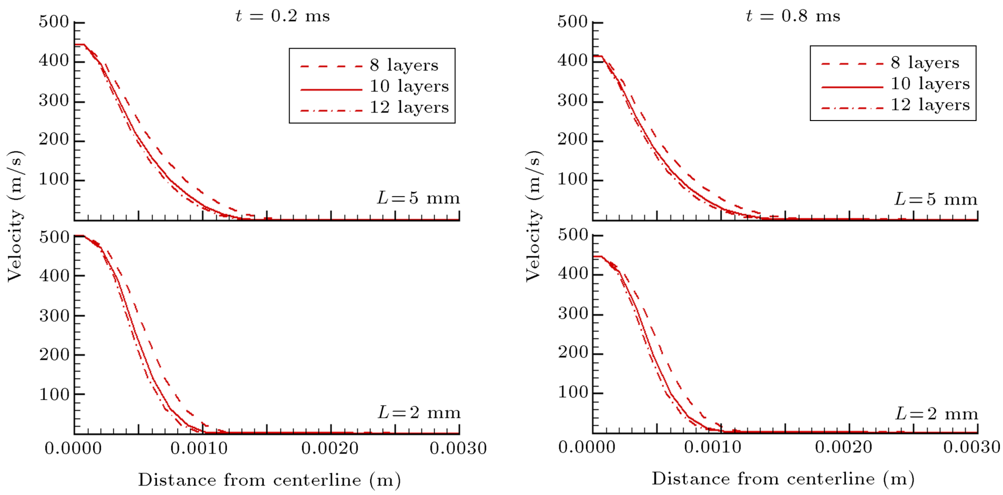


Figure 3. Velocity distribution across the jet for different locations and timings.

maximum of 10 cell layers across the nozzle diameter is sufficient.

The length scale difference between the nozzle and cylinder could be to the order of 1000. In order to get a reasonable number of grid cells, it is necessary to have very fine grids near the nozzle exit and coarser grids far from the nozzle. So, it is very important to make the variations of mesh size as smoothly as possible between these two levels. It means there should not be any jump in cell size or, at least, it should be controlled at a reasonable level.

To avoid numerical instability, the so called "transition layer" should be located far from the nozzle exit. In under-expanded nozzle flow the first transition layer needs to be located at least 8 $\sim$ 10 times the nozzle diameter downstream from the nozzle. A graphical representation of the grid in the near field and the transition layer is shown in Figure 4.

While the first layer of "cell size change" should be located far from the nozzle, other transition layers for successive coarsening of the mesh have less influence on numerical instability in comparison to the first transition layer.

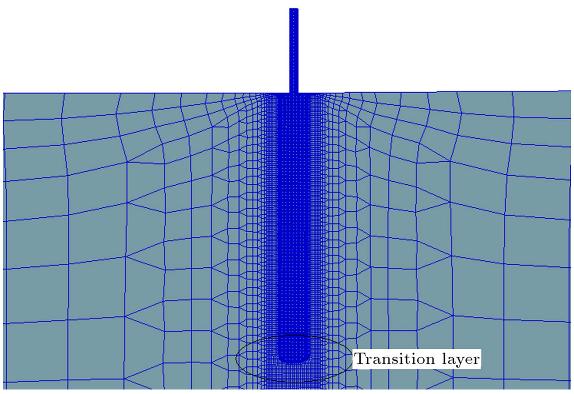


Figure 4. Location of transition layer and inlet boundary with respect to nozzle exit.

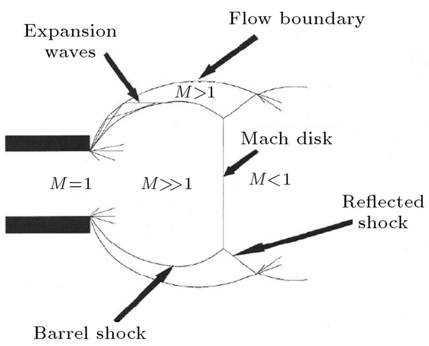
On the other hand, the inlet boundary should be located far upstream from the nozzle exit to avoid a critical condition occurring at the inlet boundary. This condition is more realized in under-expanded nozzles.

Challenges, like those mentioned above, are common in the solution process and necessitate detailed validation of the numerical model in terms of grid generation, grid independency and numerical instability prevention. The model should also be able to predict the flow pattern inside the cylinder. To assure the model accuracy and robustness in injection and flow calculations, three cases of validation have been performed based on available literature data. The first case is transient methane injection into a chamber of constant volume. The second case is air injection into a chamber of constant volume with wall impingement. Finally, a real engine-like time-dependent methane injection into a chamber of constant volume has been taken into consideration.

3.1. Validation; Case 1

A constant volume chamber with a centrally mounted injector, similar to the case of Ouellette and Hill [11], has been used to build up a numerical model and evaluate its basic characteristics. A total number of about 380000 hexahedral/hybrid grid cells have been generated to predict the flow accurately. Three transition layers have been used to coarsen the mesh inside the chamber. The number of mesh refinement levels and their locations has been obtained based on the settings in the grid independency check. The injection conditions are listed in Table 3. The boundary conditions are set, based on the data listed below.

The numerical model has been developed in AVL FIRE software using given conditions, and the injection modeling of methane gas into air has been undertaken. As the model applies to both subsonic and sonic nozzle exits, the results have been used to study injection



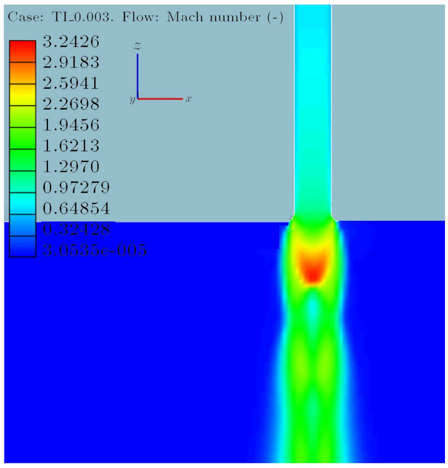


Figure 5. Flow pattern in under-expanded flow: (a) A graphical representation [11]; and (b) calculated.

Table 3. The injection conditions for the first validation case [11].

Chamber dimensions	Radius: 20 mm, Length: 90 mm
Injection pressure and temperature	$P_{0,\text{inj}} = 15$ MPa, $T_{0,\text{inj}} = 350$ K
Chamber pressure and temperature	$P_{0,\text{ch}} = 5$ MPa, $T_{0,\text{ch}} = 850$ K
Wall temperature	$T_W = 450$ K
Nozzle diameter	$d_N = 0.5$ mm
Injected mass	3.5 mg
Turbulence level	1.5 m^2/s^2

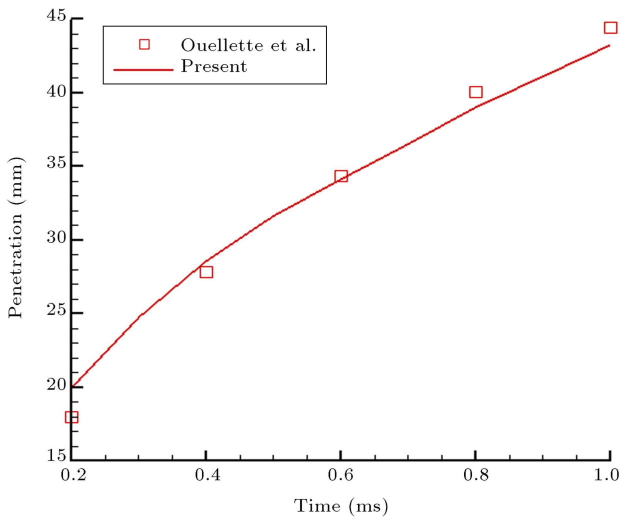


Figure 6. Comparison of jet penetration for first validation case.

in under-expanded flows, which is the case in high pressure gas injection. A snapshot of the flow field near the nozzle exit is shown in Figure 5(b). One can easily see the complex pattern of flow with a mach number of about 1.0 at the inlet boundary and a maximum mach number of 3.24 at about one millimeter downstream of the nozzle exit.

For convergent nozzles, there is a specific pressure ratio at which the flow at the nozzle exit becomes sonic. If the pressure ratio is increased even more, the flow at the nozzle remains sonic, and expansion of the jet continues downstream from the nozzle exit into the cylinder until it finally reaches the cylinder pressure. This results in a very complex flow pattern in the near field of the nozzle exit. The flow field captured agrees with the graphical representation presented in Ouellette and Hill [11], as shown in Figure 5(a).

It has been found that a minimum of eight times the nozzle diameter, i.e. 4 mm in this case, is necessary for the injector length to avoid critical conditions at the inlet in under-expanded flow cases. The first transition layer is also set to be located at least 10 times the nozzle diameter downstream from the nozzle exit to prevent numerical instability in the solution.

The results for jet tip penetration along the injection axis have been compared in these cases. Figure 6 shows such a comparison. As seen, there is very good agreement between the results. This indicates proper adjustment of numerical model parameters to predict injection penetration and flow field inside the cylinder. In the first time steps, there is a slight difference between results, which could be described, regarding the difference between numbers of grid points across the nozzle diameter. In the present study, the number of grid points across the nozzle diameter has been set to at least 10 cell layers, which introduce less numerical diffusion and, thus, more jet penetration estimation.

3.2. Validation; Case 2

As natural gas is injected at elevated pressures into the cylinder through a very small diameter nozzle, high velocities are experienced during the course of the injection. On the other side, to obtain a stratified mixture, the SOI is retarded, i.e. at crank angles where the piston is moving upwards and the distance between the piston head and the nozzle exit becomes less and less. In such a case, impingement of the jet to the piston head is inevitable. The impingement of the jet to the piston head affects the jet shape and jet induced flow field inside the cylinder. It also makes a radial motion of flow towards the cylinder walls. This can make the flow quite different in comparison to injection of gaseous fuel into very large or open environments. So, it would be necessary to study the effects of gas jet impingement on the walls. Furthermore, the numerical models ability to correctly predict jet penetration into the impingement area should be studied. In this regard, the numerical study of Andreassi et al. [16] and the experimental study of Tomita et al. [21], with air injection, have been considered. Nearly 450000 grid cells were generated. 10 grid cells in the nozzle throat have been used and grids are coarsened in 3 stages downstream of the injector nozzle. The injection conditions in this case are shown in Table 4.

Based on the given condition in Table 4, a numerical model has been developed using AVL FIRE software, and the injection and impingement of air into a chamber filled with air has been modeled. The result of air jet penetration across the plate is shown in Figure 7. As readily seen from the figure, there is excellent agreement between the results.

3.3. Validation; Case 3

The injection process under real conditions of a direct injection CNG engine is a strongly transient process. Regarding needle movements in injection shots, the numerical model should be able to address highly transient pressure waves inside the injector and cylinder. This is a slightly different case of injection, because, in the injection process, the needle might bounce and, thus, a more complicated flow pattern is formed in comparison to a simple case of transient injection.

Table 4. The modeling conditions for the second validation case [18].

Injector diameter	0.82 mm
Mean injection pressure P_{inj}	4.94 bar
Injection temperature	298 K
Environment temperature	295 K
Environment pressure	1 bar
Environment fluid	Air
Wall distance from nozzle	15.5 mm
Injected fluid	Air
Re	60000
Chamber dimensions	100*100*15.5 mm

In order to check the developed numerical model in a real injection process, it was decided to perform direct methane injection modeling based on the setup in [1]. In that study, the injection had been done into a pressure chamber with simple cubic geometry of 200 mm in the edge length. The chamber is filled with air. The modeling conditions are presented in Table 2. A total of approximately 290000 computational cells have been generated.

The case has been developed in AVL FIRE, and methane jet penetration into the media has been measured. Figure 8 shows a comparison of jet penetration downstream from the nozzle. It can be seen that there is excellent agreement between the experimental and numerical results.

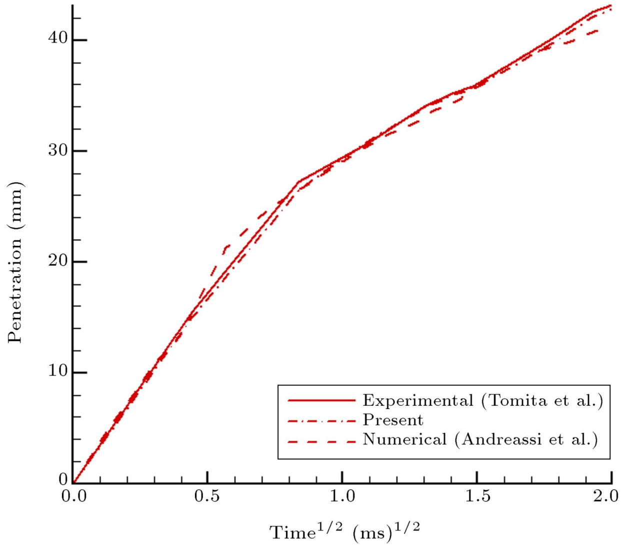


Figure 7. Comparison of jet penetration for different cases in second validation case.

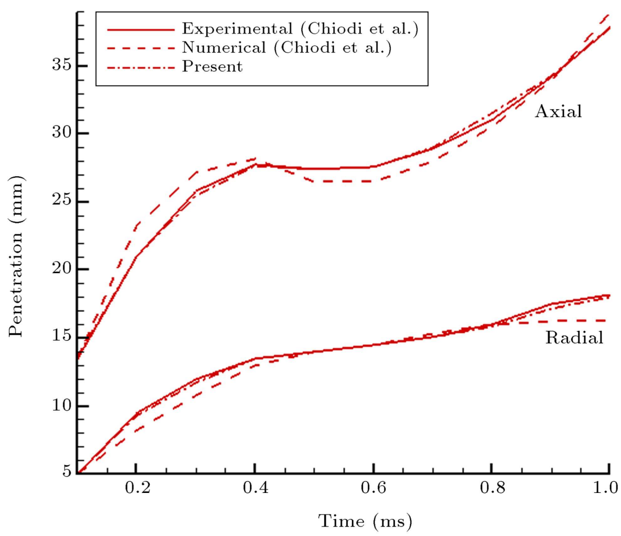


Figure 8. Comparison of jet penetration in axial and radial directions for third validation case.

4. Model application and results

The 3-D validated numerical model has been applied to the simulation of methane direct injection into different engine combustion chambers with real-like moving pistons. In this context, the effect of five piston-cylinder head configurations has been studied on the mixture homogeneity and distribution inside the cylinder. An inwardly-opening single-hole injector with a centrally mounted location has been adapted to all cases. The objective was to determine a suitable configuration for the piston head geometry with regard to injector location. Table 5 summarizes the investigated combustion chamber geometries, and a view of the geometries can be seen in Figure 9.

The base engine has a pent-roof cylinder head configuration, and each piston design should be adapted with its suitable combustion chamber geometry. However, to focus only on injection and piston geometry, a flat cylinder head has been used for all cases in this step. Also, in this phase of study, the compression ratio has been kept constant at 9.5, in all cases, which is the same as in the base engine.

The grid generation process was performed resulting in approximately 200000 cells for different cases. This value is related to the maximum number of cells in each case, i.e. the piston at BDC. In all cases, the selection of the number of grid cells, and the strategies for coarsening the mesh in the cylinder volume have been performed based on the settings of validation cases. In Figure 10, a view of the generated grid for a narrow bowl piston head type is shown.

Table 5. Main characteristics of selected combustion chambers.

Piston configuration	Bowl diam. (mm)	Bowl depth (mm)
Flat	-	-
Base	65	4
Narrow bowl	38	10
Large bowl	59	5
Large bowl modified	59 (fillet)	5

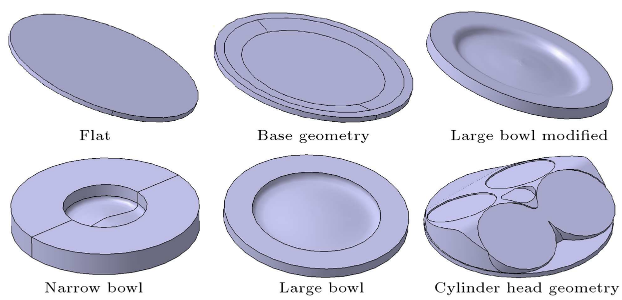


Figure 9. A view of combustion chamber and cylinder head geometries.

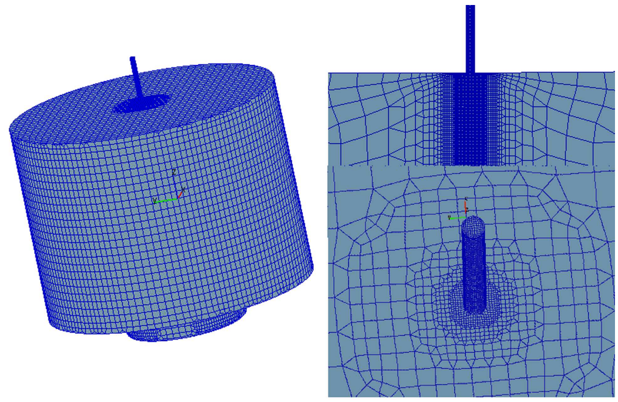


Figure 10. A view of computational grid for narrow bowl geometry with single-hole injector: (a) A whole view; and (b) injector area.

The total injected fuel has been set to form a generally lean mixture of air and fuel inside the cylinder after EOI. The injector and cylinder are co-axial, i.e. the injection axis is the same as the cylinder axis. Injection duration has been kept constant in all cases to 90 degrees of crank angle. The effect of injector needle movement has been accounted for by application of variable fuel rates under inlet boundary conditions with a maximum fuel rate of 2.1 grams per second at 2000 RPM. The Start Of Injection (SOI) timing for all simulated cases is 120° $\mathrm{BTDC_{firing}}$. The chamber pressure at the start of the calculation is set for standard air conditions. The engine speed is considered to be 2000 RPM in all cases.

In Figure 11, the effect of different piston head shapes on mixture formation and jet development inside the cylinder has been shown through some snapshots of flow at 40° $\mathrm{BTDC_{firing}}$. The equivalence ratio has been chosen to represent distribution of the air-fuel mixture in the cylinder.

As seen in the figures, a flat piston shape causes the flow to be driven to the cylinder edges. But, a narrow bowl combustion chamber causes the flow to recirculate and generate a kind of tumble motion inside the cylinder, resulting in a narrower jet shape and more stratification. It has been found that even small changes, such as rounding up the edge of the combustion chamber, can have a major effect on fuel jet shape inside the cylinder. Despite the difference in flow field, the large bowl and large bowl-modified combustion chambers showed, generally, similar characteristics to the base engine geometry. All these cases have been realized to develop a wide and weak tumble effect in the combustion chamber, resulting in lower stratification of in-cylinder charge. In contrary to flat combustion chambers, these shapes tend to avoid the jet from penetration towards the cylinder edges and, thus, make a more compact jet shape.

Regarding the valve timing for the base engine,

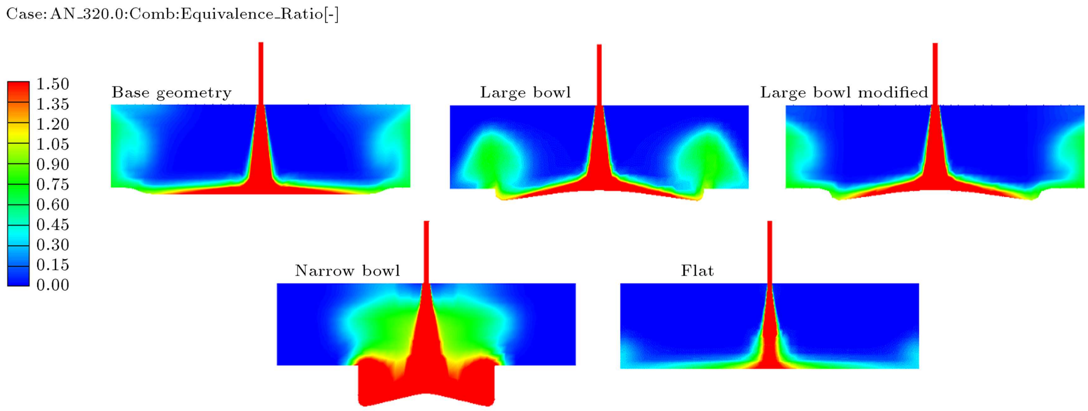


Figure 11. Effect of combustion chamber geometry on jet shape inside cylinder on a planar section through injector axis; all cases at 40° BTDC.

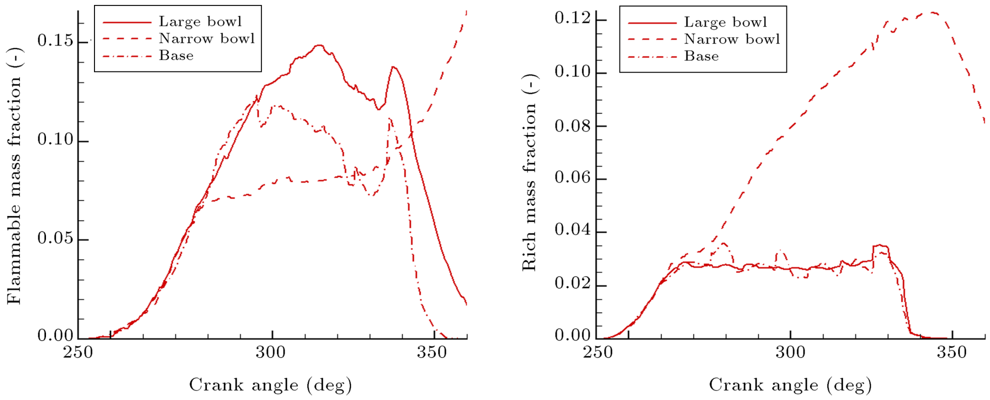


Figure 12. Temporal variations of mixture characteristics for different combustion chamber geometries: (a) Flammable mass fraction; and (b) rich mass fraction.

a SOI timing of 120° $\text{BTDC}_{\text{firing}}$ gives the completely stratified operation of the engine. For such conditions, there is an optimum SOI timing. For more retarded SOI timing, there is not enough time for mixing. On the other hand, further advancing the SOI might decrease volumetric efficiency. These effects are to be studied further.

One other reason to narrow the SOI timing gap has been realized, regarding the mixing of air and fuel after injection. The earlier SOI timing provides more time for the fuel to mix with the air before ignition timing. This might lead to the over-mixing of air-fuel, resulting in a more-than-necessary lean mixture and poor stratification. The mixture in such conditions might be too lean to have combustion after ignition.

In order to get a better understanding of mixture preparation inside the cylinder, a quantitative comparison could be easily done based on flammable mixture definition. As stated earlier, the flammability limits of 0.8 and 1.4 on RAFR have been considered for stable combustion, based on the literature. A flammable mass fraction inside the cylinder could be obtained based on these limits. The flammable mass fraction is simply the fraction of the flammable mass of the mixture to the total mass of the mixture. Similarly, a rich mixture mass fraction is defined as the ratio of the rich mixture mass to the total mass of the mixture.

The temporal evolution of the flammable mass fraction for different combustion chamber geometries is shown in Figure 12(a). To make comparisons easier, three different geometries have been considered: the narrow bowl, the large bowl and the base geometry. It can be seen that as more fuel is injected into the cylinder, the flammable mass fraction shows an increase in all cases. This value shows a maximum in the case of large bowl and base geometries, and a decrease in the level of flammable mass fraction is experienced after that point. The reason for such a decrease could be related to the motion of fuel towards the cylinder walls. Such a motion can be observed in the fuel distribution contours in Figure 11. After the EOI timing, the flammable mixture in these two cases rapidly decreases to near zero values, which shows over the mixing of fuel and air. Unlike these two geometries, the flammable mass fraction in the narrow bowl geometry shows a constant increase. This trend is more realized near the EOI timing at which a rapid increase in flammable mass fraction occurs.

Such an increase is easily seen in the rich mass fraction in Figure 12(b) too. In this figure, the narrow bowl geometry case shows completely different characteristics in comparison to the other two cases. Rapid increase of rich mass fraction near EOI timing for the case of narrow bowl geometry is quite opposite to the decrease in rich mass fraction in other cases. Again, the two other cases are found to behave similar to each other in rich mass fraction variations. The initial increase in rich mass fraction in all cases is related to the initial fuel injection phases. The mixing of fuel with air in outer regions of the cylinder causes both flammable and rich mass fractions to decrease to near zero in large bowl and base geometry cases. But, as the narrow bowl geometry tends to keep the whole mixture in a "trapped" region for a while, both flammable and rich mass fractions show a rapid increase in the case of narrow bowl geometry. Finally, the mixing between the trapped region and outside air causes the rich mixture to decrease after EOI timing, in this case.

The difference in results can be explained regarding the effect of combustion chamber geometry on the flow. More trapped mixture inside the bowl of the narrow bowl geometry is the primary reason for the higher flammable and rich mass fractions compared to other two cases.

To sum up, the presented results show that, generally, narrower combustion chamber geometries promise better characteristics for mixture stratification. However, the accurate matching of injection timing, combustion chamber geometry and injector location is necessary to get better combustion characteristics for the engine. Such a matching requires extensive detailed studies regarding optimized combustion chamber geometry and combustion processes. A further assessment of more complex combustion chamber geometry and combustion studies will be presented in future publications.

As may be inferred from the presented results, flat piston combustion chamber designs do not seem to be appropriate for charge stratification. In this case, the mixture is spread widely into the chamber. Such a characteristic is more appropriate to the homogenous charge combustion system with advanced SOI timing. This is better realized regarding more time available for the mixture preparation with advanced SOI timing.

When a bowl-in-piston combustion chamber is used, the injected jet experiences a recirculation motion towards the cylinder head and, thus, the injected gas forms a compact jet shape. Generally, narrower bowls result in more compact jet shapes. Of course, there are other variables that affect the jet shape inside the cylinder, namely, injector type, bend angle of injector nozzle and jet targeting. Only by matching these variables can one obtain a compact flammable mixture inside the cylinder.

A qualitative as well as quantitative representation of the results shows that the narrow bowl piston head shape has the best stratification characteristics among all the studied piston head shapes. Using a centrally mounted inwardly-opening single-hole injector with such a piston head shape tends to create a narrow jet near the cylinder axis. This position is important regarding spark plug location. Such a configuration shows great potential for optimizing the flow field inside the cylinder, too.

5. Cylinder head effect

In order to better understand the effects of both cylinder head and piston head on an in-cylinder flow field, a part of this study has been dedicated to the cylinder head effect. In this regard, the pentroof cylinder head shape of the base engine geometry has been taken into consideration. In this part, a flat piston head shape has been adapted in order to determine the effect of cylinder head on the in cylinder flow field. Afterwards, the effect of a narrow bowl piston head is also studied.

A maximum of about 200000 grid cells has been generated in different cases. As in the previous part, RAFR has been considered the main variable to study the distribution of the flammable fuel-air mixture inside the cylinder.

The modeling conditions have been considered exactly similar to previous cases. Injector locations are set to be centrally mounted and the engine RPM is 2000 rev/min in all cases. Figure 13 shows some snapshots of the gas jets emerged from the injector. The snapshots are taken on two perpendicular planes crossing the cylinder axis. As seen from the figures, the cylinder head shape makes the flow field quite different in comparison to the flat cylinder head cases; especially near the end of the compression stroke. In other words, the cylinder head shape has shown its greater effects on the flow field towards the end of the compression stroke, which is the most important time in terms of injection, ignition and combustion. In the early phases of injection, the cylinder head shape has little effect on jet development inside the cylinder.

The asymmetric shape of the combustion chamber, in this case, makes the flow field inside the chamber non-symmetrical. So, the flow field pattern in different sections of the combustion chamber is quite different. The non-symmetric flow field inside the cylinder makes the combustion and flame propagation inside the cylinder into a very complex asymmetrical pattern.

The pentroof effect of the cylinder head makes the flow directed towards the center of the cylinder near the end of the compression stroke. Such a flow is not experienced in flat cylinder head cases. A flat piston

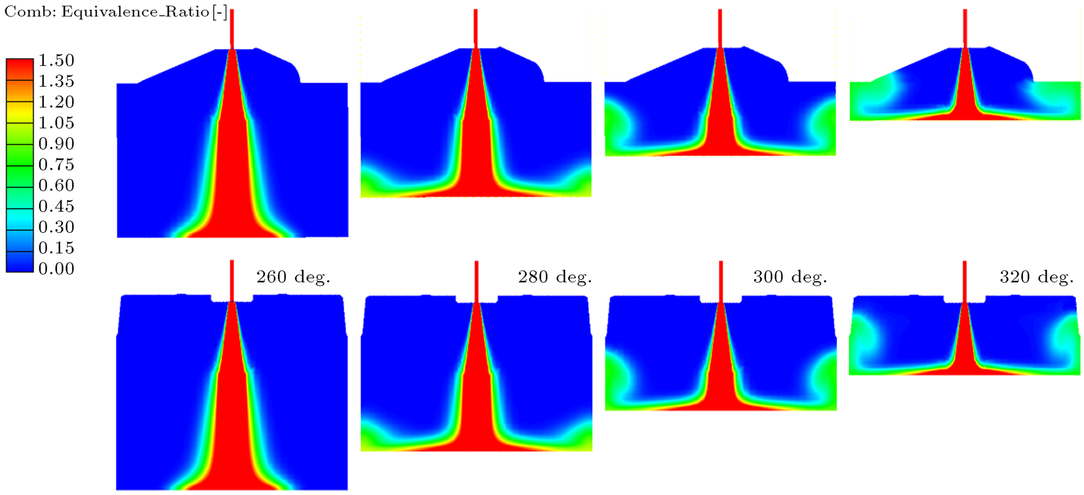


Figure 13. Comparison of flow field inside cylinder for flat piston head and pentroof cylinder head on two perpendicular planes crossing the cylinder axis.

head still tends to "flatten" the jet and direct it to the cylinder walls.

It can easily be seen in the figures that the flammable mixture is not directed towards the center of the cylinder, even near the end of the compression stroke. This might turn into a problem in terms of combustion initiation and stability, regarding the base engine spark plug location. Combustion simulations could exactly determine whether or not such a phenomenon occurs. More studies will be performed on the subject.

Spark plug location is one of the most important variables which control the combustion inside the cylinder of spark ignition engines. An asymmetrical location for the spark plug leads to non-symmetrical flame propagation inside the cylinder. This could lead to asymmetrical pressure distribution on the piston head, which could create some problems. So, it is intended, as much as possible, to set the combustion chamber configuration to have symmetrical combustion inside the cylinder.

Thus, to reduce the effects of non-symmetrical combustion inside a cylinder, a near center spark plug location has been adopted in many combustion chamber designs, as in the base engine. Asymmetrical gas jet development inside the cylinder leads to a poor quality mixture near the center of the cylinder at the end of the compression stroke, which is an approximate time for ignition. Unstable combustion could also be the case in such conditions. So, there is a need for a combustion chamber design which has the characteristic of directing gas jet flow towards the center of the cylinder, i.e. a narrow bowl piston head shape.

It should be noted that the gas jet emerged from this kind of injector tends to make a very narrow stratified charge mixture inside the cylinder. As the jet targeting in single-hole injector cases is central, the gas jet is concentrated near the cylinder axis. If suitably controlled, this kind of jet could lead to a stratified charge mixture preparation which could still develop a flammable mixture near the center of the cylinder by the end of the compression stroke at approximate ignition time. So, a case of a single-hole injector with a pentroof cylinder head and narrow bowl piston head shape has been taken into consideration as a good option for combustion chamber configuration.

In this case, another numerical model has been constructed in AVL FIRE, and a numerical investigation of flow has been performed under previously mentioned conditions. In Figure 14, some snapshots of flow inside a cylinder have been shown. It can be seen from the figure that flow field and jet shape development are different from those in the flat piston head case. In this case, flow is mainly restricted to the central space of the cylinder. In the last frame, it is clearly shown that a flammable mixture is present near the cylinder centerline.

In such a case, the piston bowl, in combination with the cylinder head shape, provides a near center space for the mixture to develop. Such a configuration forces the flammable mixture to move towards the center of the cylinder, which is mainly the spark plug location.

It can be seen from the figure that a narrow bowl piston head shape could help reduce the effect of poor mixture quality near the spark plug location. As shown in Figure 14, the flammable mixture is present near the cylinder axis on both displayed sections. This suggests that a narrow bowl combustion chamber design is more suited to the pentroof cylinder head.

In order to better investigate the effect of different configurations of a cylinder head-piston head, temporal variations of flammable and rich mass fractions for

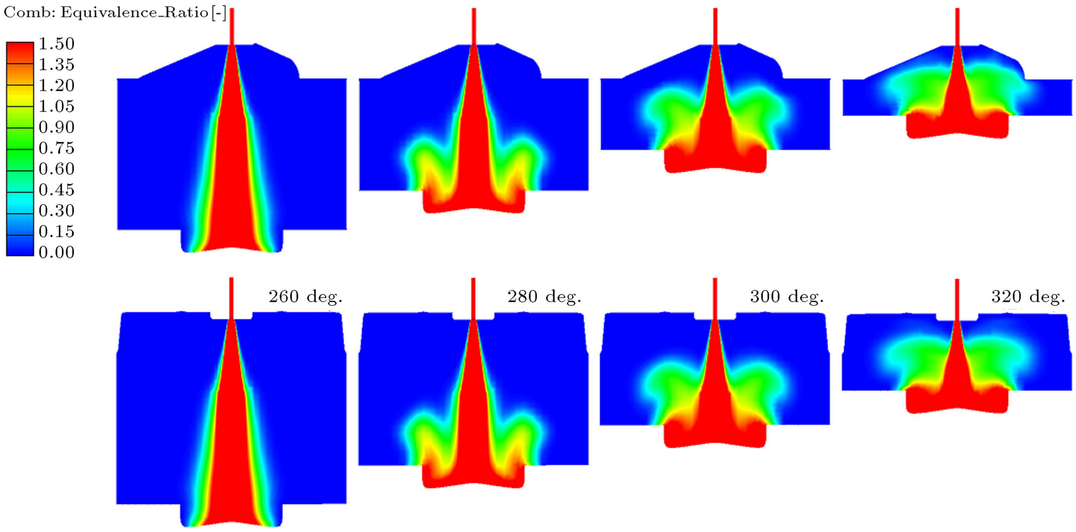


Figure 14. Comparison of flow field inside cylinder for narrow bowl piston head and pentroof cylinder head on two perpendicular planes crossing the cylinder axis.

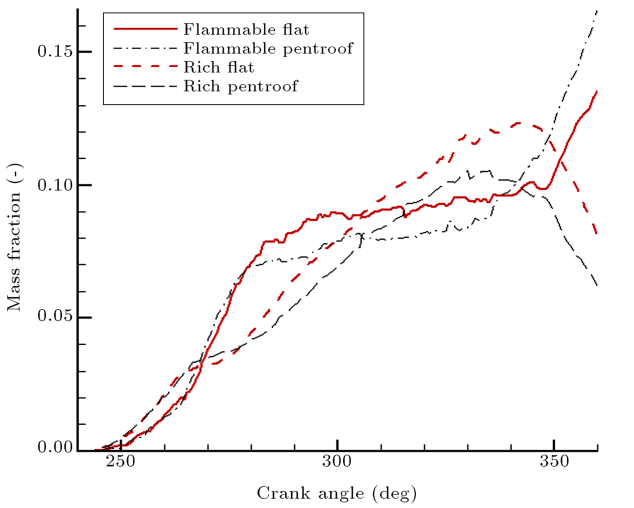


Figure 15. A comparison of flammable and rich mass fractions for flat and narrow bowl piston head shapes with pentroof cylinder head.

these cases have been depicted in Figure 15. As observed, the combination of a pentroof cylinder head with a narrow bowl piston head exhibits higher values of flammable mass fraction in comparison to the flat piston head shape. The value of the rich mass fraction is lower for narrow bowl cases, accordingly.

Quantitative and qualitative representation of the flow field inside the cylinder in these cases shows that the pentroof-narrow bowl configuration of cylinder head and piston geometries shows the best characteristics in terms of mixture quality and stratifications. Of course, the current configuration of the piston head shape in conjunction with the pentroof cylinder head and single-hole injector needs some further modification to enhance the flow field inside the cylinder.

Other justifications, such as compression ratio adjustment and inlet/exhaust ports, are needed to complete the modification of the combustion chamber. In this regard, combustion and open cycle simulations should be performed to determine whether or not the configuration is suitable for CNG DI performance or not. More studies are to be undertaken on this subject and the results will be presented in future publications.

6. Conclusions

Based on the discussions presented in the previous sections, the main conclusions are summarized as follows:

- The modeling results are strongly sensitive to the number of grid points across the nozzle diameter; boundary condition locations and boundary values seem to play a critical role. Proper setting of grid generation parameters and boundary conditions is necessary to avoid numerical instability.
- A minimum of 10 cell layers across the injector nozzle is needed to predict the flow accurately and avoid numerical instability. Also, the transition layer should be located at least 8 $\sim$ 10 times the nozzle diameter downstream from the nozzle to avoid numerical instability.
- The modeling results are shown to have very good agreement with validated experimental and numerical results from literature.
- Four axisymmetric bowl-in-piston configurations along with a flat piston have been chosen to study the effect of piston head geometry. Aiming to get a stratified charge inside the cylinder, the narrow bowl configuration is shown to be a better choice because of better mixture distribution near ignition time.

- Qualitative and quantitative representation of results show that even a small difference in combustion chamber geometry can cause quite different jet shapes and mixture properties inside the cylinder.
- It has been shown that, generally, wider bowl geometries tend to spread the mixture towards the cylinder walls, while narrower bowls tend to "trap" the fuel jet in a compact area.
- Keeping the flammable mixture in a compact area seems to be an important characteristic for stratified charge operation. Thus, those geometries which tend to spread the mixture do not seem to be suitable for stratified charge operation. Further studies regarding combustion could help to determine suitable configurations.
- Narrow bowl geometry seems to have a completely different effect on the temporal evolution of flammable and rich mass fractions inside a cylinder in comparison to all other cases. Such effects are considered to be related to more mixture stratification.
- The pentroof cylinder head shape was found to create an asymmetrical flow inside the cylinder. The effect of the cylinder head is more realized towards the end of the compression stroke.
- A combination of pentroof cylinder and narrow bowl piston head shapes has been shown to be a suitable configuration in order to get a stratified charge mixture inside the cylinder. However, more modification on the flow field seems to be needed. More studies will be conducted in this case.

Acknowledgments

The authors would like to acknowledge the AVL Company for preparing FIRE software and support.

List of abbreviations

2D	Two Dimensional
3D	Three Dimensional
ASI	After the Start of Injection
BDC	Bottom Dead Center
BTDC	Before Top Dead Center
CNG	Compressed Natural Gas
DI	Direct Injection
EOI	End Of Injection
MPFI	Multi-point Port Fuel Injection
PM	Particulate Matter
RAFR	Relative Air-Fuel Ratio
RPM	Revolution Per Minute
SCRE	Single Cylinder Research Engine
SI	Spark Ignition
SOI	Start Of Injection

References

1. Chiodi, M., Berner, H.J. and Bargende, M. "Investigation on different injection strategies in a direct-injected turbocharged CNG-engine", SAE paper 2006-01-3000 (2006).
2. Li, Y., Kirkpatrick, A., Mitchell, C. and Willson, B. "Characteristic and computational fluid dynamics modeling of high-pressure gas jet injection", *ASME Trans., J of Eng. for Gas Turb. and Power*, **126**, pp. 192-197 (2004).
3. Huang, Z., Shiga, S., Ueda, T., Nakamura, H., Ishima, T., Obokata, T. et al. "Visualization study of natural gas direct injection combustion", *Proc. Inst. Mech. Eng. Part D: J Auto. Eng.*, **217**, pp. 667-673 (2003).
4. Huang, Z., Shiga, S., Ueda, T., Nakamura, H., Ishima, T., Obokata, T., Tsue, M. and Kono, M. "Basic characteristics of direct injection combustion fuelled with compressed natural gas and gasoline using a rapid compression machine", *Proc. Inst. Mech. Eng. Part D: J Auto. Eng.*, **217**, pp. 1031-1038 (2003).
5. Huang, Z., Shiga, S., Ueda, T., Nakamura, H., Ishima, T., Obokata, T., Tsue, M. and Kono, M. "Correlation of ignitability with injection timing for direct injection combustion fuelled with compressed natural gas and gasoline", *Proc. Inst. Mech. Eng. Part D: J Auto. Eng.*, **217**, pp. 499-506 (2003).
6. Zeng, K., Huang, Z., Liu, B., Liu, L., Jiang, D., Ren, Y., Wang, J. "Combustion characteristics of a direct-injection natural gas engine under various fuel injection timings", *J. App. Therm. Eng.*, **26**, pp. 806-813 (2006).
7. Papageorgakis, G. and Assanis, D.N. "Optimizing gaseous fuel-air mixing in direct injection engines using an RNG based k-ε model", *Trans. of SAE, J. of Engines*, **107**, pp. 82-107 (1999).
8. Han, Z. and Reitz, R.D. "Turbulence modeling of internal combustion engines using RNG k-ε models", *Comb. Sci. & Tech.*, **106**, pp. 267-295 (1995).
9. Li, G., Ouelette, P., Dumitrescu S. and Hill P.G. "Optimization study of pilot-ignited natural gas direct-injection in diesel engines", SAE Paper No. 1999-01-3556 (1999).
10. Abraham, J. and Magi, V. "Computation of transient jets: RNG k-ε model versus standard k-ε model", SAE Paper No. 970885 (1997).
11. Ouellette, P. and Hill, P.G. "Turbulent transient gas injections", *ASME Trans., J of Fluid Eng.*, **122**, pp. 743-753 (2000).
12. Abraham, J. "What is adequate resolution in the numerical computations of transient jets?", SAE Paper No. 970051 (1997).
13. Mather, D.K and Reitz, R.D. "Modeling the effects of auxiliary gas injection on diesel engine combustion and emissions", *Trans. of SAE, J. of Engines*, **109**, pp. 443-458 (2002).

14. Baratta, M., Catania, E. and Pesce, F.C. "Multidimensional modeling of natural gas jet and mixture formation in DI SI engines- Development and validation of a virtual nozzle model", *ASME proc. of the Int. Comb. Eng. Div. Tech. Conf. ICES 2009*, Milwaukee, Wisconsin, USA, pp. 583-596 (2009).

15. Baratta, M., Catania, A.E. and Pesce, F.C. "Multidimensional modeling of natural gas jet and mixture formation in direct injection spark ignition engines - Development and validation of a virtual injector model", *ASME Trans., J. of Fluid Eng.*, **133**, pp. 041304-1 - 041304-14 (2011).

16. Andreassi, L., Facci, A.I., Krastev, V.K. and Ubertini, S. "Multidimensional modeling of gaseous injection: Analysis of an impinging jet", *Int. J. Heat and Fluid Flow*, **31**, pp. 909-915 (2010).

17. Baratta, M., Catania, E., Spessa, E., Herrmann, L. and Roessler, K. "Multi-dimensional modeling of direct natural-gas injection and mixture formation in a stratified-charge SI engine with centrally mounted injector", SAE paper 2008-01-0975 (2008).

18. Kim, G.H., Kirkpatrick, A. and Mitchell, C. "Computational modeling of natural gas injection in a large bore engine", *ASME Trans., J. of Eng. for Gas Turb. and Power*, **126**, pp. 656-664 (2004).

19. Yadollahi, B. and Boroomand, M. "The effect of piston head geometry on natural gas direct injection and mixture formation in a SI engine with centrally mounted single-hole injector", SAE paper 2011-01-2448 (2011).

20. Yadollahi, B. and Boroomand, M. "A numerical investigation of combustion chamber geometry effects on natural gas direct injection properties in a SI engine with centrally mounted multi-hole injector", *ASME Proc. of the Int. Comb. Eng. Div. Tech. Conf. ICES 2012*, Torino, Italy, pp. 797-808 (2012).

21. Tomita, E., Hamamoto, Y., Tsutsumi, H., Takasaki, S., Watanabe, T. and Yoshiyama, T. "Visualization of ambient air motion and entrainment into a transient gas jet impinging on a flat wall", SAE paper 952513 (1995).

Biographies

Bijan Yadollahi received his BS degree in Mechanical Engineering from the University of Shiraz, Iran, in 2003, and MS and PhD degrees from Amirkabir University of Technology, Tehran, Iran, in 2006 and 2013, respectively. His research interests include internal combustion engines and CFD.

Masoud Boroomand was born in Abadan, Iran, in 1958. He received his BS degree from Abadan Institute of Technology, Iran, his MS degree in Applied Energy from Birmingham University, UK, in 1987, and his PhD degree in Turbomachinery from Cranfield University, UK, in 1992. He is currently Associate Professor in the Aerospace Engineering, Department of Amirkabir University of Technology, Tehran, Iran. His research interests include turbomachinery and gas turbines.