Fuel 107 (2013) 52–62

Contents lists available at SciVerse ScienceDirect

Fuel

journal homepage: www.elsevier.com/locate/fuel

The effect of combustion chamber geometry on injection and mixture preparation in a CNG direct injection SI engine

B. Yadollahi, M. Boroomand *

Amirkabir University of Technology (Tehran Polytechnic), Tehran, Iran

HIGHLIGHTS

- ▶ The results show strong sensitivity to grid size across the nozzle diameter.
- ▶ Narrow bowl configuration showed to be best choice for charge stratification.
- ▶ Single-hole and multi-hole injectors exhibit positive characteristics for the flow.
- ▶ The influence of pentroof cylinder head occur towards the end of compression stroke.
- ▶ The pentroof cylinder head also forces a non-symmetrical flow field inside cylinder.

ARTICLE INFO

Article history:
Received 28 November 2011
Received in revised form 25 December 2012
Accepted 2 January 2013
Available online 29 January 2013

Keywords:
CNG direct injection
Spark ignition
Mixture preparation
DISI
High pressure injection

ABSTRACT

In pursuing a project to convert a four cylinder gasoline multi-point port fuel injection engine to a CNG direct injection engine, a numerical model has been developed in AVL FIRE software to undertake a detailed numerical investigation on the effects of combustion chamber geometry in such engines. Two main phases have been considered in the present study. In the first phase, aiming to fully investigate flow field and mixing process, multi-dimensional numerical modeling of transient gas injection has been performed. In order to verify the accuracy of the model, two different validation cases have been employed. The results showed that the models are quite capable of grasping all the significant physical phenomena in the process. Adoption of such a modeling was found to be a challenging task because of required computational effort and numerical instabilities. In all validation cases, experimental and numerical results were observed to have excellent agreement with each other.

In the second phase, using the moving mesh capability the validated model has been developed to include methane direct injection into the cylinder of a direct injection engine for various combustion chamber geometries. Five different piston head shapes along with two injector types of single hole and multi hole have been taken into consideration. A centrally mounted injector location has been adapted to all cases. The effects of combustion chamber geometry, injection parameters, injector type and cylinder head shape have been studied on mixing of air–fuel inside cylinder. Based on the results, suitable geometrical configuration for the new NG DI engine has been discussed.

1. Introduction

It is well known that solving three dimensional compressible turbulent reacting flows in an internal combustion engine is a challenging task. The situation gets more complex if direct injection of fuel is considered, and it will be even more complex if a gaseous fuel such as NG is targeted. In fact, mixture formation with natural gas due to its low density is more critical than in liquid fuel engines. In such cases 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 in comparison to gasoline engines [1].

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

Abbreviations: 2D, two dimensional; 3D, three dimensional; 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.

* Corresponding author. Address: Department of Aerospace Engineering, Amirkabir University of Technology, Tehran Polytechnic, Tehran, Iran. Tel.: +98 2164543280; fax: +98 2166959020.

E-mail address: Boromand@aut.ac.ir (M. Boroomand).

0016-2361/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.fuel.2013.01.004

Very high pressure ratios between fuel rail and in-cylinder, causes the flow at injector exit to be typically under-expanded. It means that the flow becomes sonic at nozzle exit and undergoes a complex pattern of shock-expansion waves downstream from the nozzle exit. It has been shown that accurate capturing of such a pattern in the near field strongly affects the complete jet shape and mixture formation downstream from the nozzle [2].

Ouellette and Hill [3] have studied turbulent transient methane and air injection into a constant volume chamber. A multi-dimensional numerical modeling as well as experimental investigation has been performed on subsonic and sonic jets emerging from a single nozzle into the chamber. The KIVA code has been used in the study to perform modeling. The results for jet shape and penetration have been presented in different cases.

Flow field inside the cylinder of direct injection engines is highly influenced by the geometry of combustion chamber. Effects of piston head and cylinder head together with injector location should be studied as the main geometrical parameters. In fact, a small change in geometry can lead to considerable changes in the mixture distribution inside the cylinder. In several studies such as [1,4–6] the impact of combustion chamber geometry on in cylinder flow field and mixture preparation has been studied.

To address a few, multi-dimensional modeling with focus on turbulence modeling has been performed in [7–9] and the evolution of gas jet injection is studied in [3,7,10].

In [5,6] the effect of piston head shape has been studied on the fluid flow, combustion and emissions for gas fuelled engines. In this regard, based on results it has been concluded that bowl-in-piston configurations for piston head shape could be beneficial in natural gas fueled engines.

Due to strong influence of geometry on flow field, optimum flow field and jet development inside cylinder could not be achieved unless the injector–chamber configurations are studied in conjunction with each other. Huang et al. [11–14] conducted a set of experimental studies on different injection and combustion characteristics of natural gas in different injector–chamber configurations using a rapid compression machine and also a natural gas fuelled engine.

There are several studies that have been focused on direct injection modeling as one of the most important parts of flow simulation in an engine. Li et al. [2] have conducted a 2D numerical modeling using method of characteristics to study under-expanded gas injection into the cylinder of a large bore engine. In this study, it has been reported that main parameters determining the type of flow exiting from a sonic nozzle are the pressure expansion ratio and the nozzle geometry. It has been mentioned that accurate implementation of boundary condition at the nozzle exit is very important, because it controls the jet mixing downstream from the nozzle.

In [2] and also in [3,15] the grid density at the nozzle exit has been discussed. It has been reported that a minimum of about 10 cell layers are necessary for accurate capturing of flow field in the near field.

In [1] an idea of fictive droplets of gas has been introduced into a numerical model in Quicksim software to reduce the grid number requirements of the modeling. Also in [16] a phenomenological model has been implemented along with KIVA code to predict the flow in the near field.

A virtual nozzle approach has been used in modeling the injection instead of computing the detailed near field flow in [10,17,18]. Such cases have their inlet boundary condition set at the mach disc in nozzle exit, and thus the inside injector space is eliminated. This method results in the decrement of grid density at nozzle exit; but is limited to two dimensional slot and hole injector geometries and also it requires accurate data about mach disc size and location.

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 [4,19]. In [4] a multi-dimensional modeling approach has been undertaken using STAR-CD software. A centrally mounted outwardly-opening injector has been implemented in the model and injection and mixing has been studied in different poppet-valve chamber geometries. The injection process has also been studied in a type of single cylinder research engine.

2. Present study

In a project to convert an existing 1800 cc MPFI gasoline engine (Fig. 1) to a direct injection NG engine with minimum modifications, extensive numerical simulations have been undertaken. Multi-dimensional numerical modeling of transient injection process, mixing and flow field has been performed in this regard.

There are different aspects of flow inside cylinder which should be studied. One of the most important parts of which are combustion chamber geometry and injection study, which have been taken into consideration in present work. The objective has been to determine a suitable configuration for the piston head geometry with regard to injector location and also to choose the best injector type for the application based on available selections. In the present study all the numerical models have been developed in AVL FIRE. Table 1 illustrates the basic characteristics of the engine.

To perform modeling, two injector types of single hole and multi hole have been chosen. The multi hole injector has six equally spaced nozzles with a 30° bend angle to the axial axis. Both injectors exhibit inwardly opening needle. The injector locations have been set to lay at the axis of cylinder; i.e. the injectors are centrally-mounted.

Mixture preparation for DI engines lays into two main strategies. Firstly, a homogenous charge mixture preparation with early injection in intake stroke, and secondly, a stratified charge mixture preparation with late injection in compression stroke. The latter is reportedly a more efficient strategy in terms of fuel economy [1,4]. So it has been intended to use stratified charge mixture preparation strategy in the modifications.

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

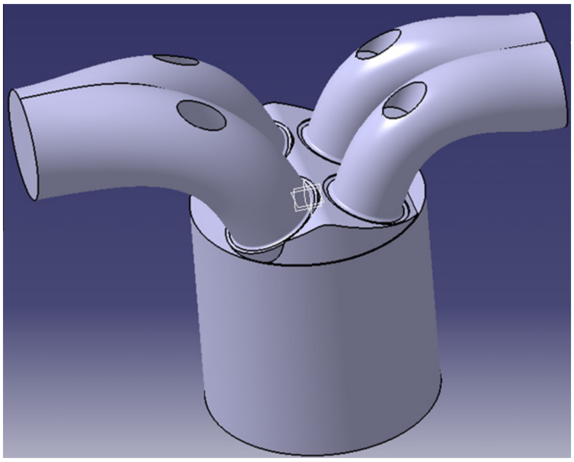

Fig. 1. A view of base engine geometry and ports.