Amirkabir Journal of Mechanical Engineering

Amirkabir J. Mech. Eng., 49(1) (2017) 13-16
DOI: 10.22060/mej.2016.729

Numerical Study of a Wall-Impinging Gaseous Jet on a Flat Plate

A. Nassiri Toosi[1*], A. HajiAliMohammadi[2], A. Heidary[1]

[1] School of Automotive Engineering, Iran University of Science and Technology, Tehran, Iran
[2] Mechanical Engineering Department, Semnan University, Semnan, Iran

ABSTRACT: In gaseous fueled direct injection internal combustion engines, in the homogeneous combustion mode, fuel-air mixture is formed by impinging fuel jet into the moving piston crown. Study of mixture formation by wall-impinging jet has a great advantage in developing and improving this kind of engines. In this study, by using ANSYS Fluent software, behavior of a wall-impinging jet on a flat plate is simulated in the atmospheric pressure and validated by using the available experimental data. By using the validated model, effects of different parameters including pressure ratio, engine speed, and injection timing on air-fuel mixture formation (Methane-Air) in a closure are studied, trying to simulate the effect of different parameters on mixture formation in an engine during its compression stroke. The results indicate that increasing the pressure ratio leads to increasing the penetration length of the injected fuel, while increasing the engine speed decreases the penetration length of the injected fuel. Additionally, injection timing may increase or decrease the penetration depth, depending on other factors including: engine speed, injection pressure, and injection duration.

Review History:

Received: 1 June 2015
Revised: 31 October 2015
Accepted: 24 January 2016
Available Online: 8 November 2016

Keywords:

Injection
Impinging jet
Penetration depth
Fuel injection

1- Introduction

One of the most important topics in DI engines researches is air-fuel mixture formation. In gaseous fueled DI engines, injection of compressed gas is done during a short period of engine operation time. Since in gas fueled DI engines, fuel is directly injected in to the combustion chamber, air-fuel mixture should be as homogeneous as possible to have an efficient combustion with reduced CO and HC emissions [1]. This issue has a great impact on engine performance, especially in stratified combustion mode.

Chinto et al. [2] numerically simulated transient methane injection in atmospheric air. Speed and temperature distribution, and viscosity of methane were studied at the bottom side of the steady jet. Craft et al. [3] used numerical methods to solve turbulent impinging jet problems. Four turbulent models were compared with each other. It was observed that eddy viscosity model results have lower agreement with experimental data, due to its' base weakness in tension-strain equation. Afroz and Sharif [4] investigated numerical heat transfer of twin oblique impinging jet with an isothermal plate. They showed that SST k-ω model gives better predictions results than k-ε. Additionally, effects of plate distance, plate angle and Reynolds number on Nusselt number were investigated.

In the current study, an analytical methodology is proposed to simulate and calculate the pressure loss during gas transient injection. Afterwards, by using ANSYS Fluent software [5], direct injection of helium jet in to atmosphere will be simulated and validated. The same model then is used to simulate direct injection of methane gas in a conventional DI engine to study the effect of different parameters on penetration length, including: engine speed, gas pressure and injection timing.

Corresponding author, E-mail: anasiri@iust.ac.ir

2- Calculation of the injected gas pressure loss during transient injection

During transient injection, pressure loss leads to lower output pressure, which decreases theoretical pressure ratio. In this work, based on shock wave tube phenomenon, an analytical model is proposed to model direct injection of the gaseous fuel. As it can be seen from Fig. 1, if the upstream and downstream of injector needle modeled as the high and low pressure sections of the shock wave tube (separated by a diaphragm), actual pressure ratio and pressure loss can be calculated during injection [6].

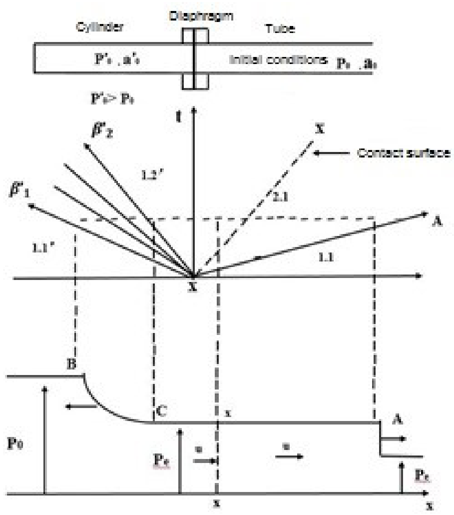

Figure 1. Different sections of a shock wave tube [6]

The rupture of diaphragm results in a moving compression wave inside the tube and an expansion wave inside the cylinder.
The pressure between the two sections (P_e) is less than initial high pressure section ($\acute{P}_o$) and more than initial high pressure section (P_o). If the injector needle movement and its opening are assumed to be similar to diaphragm rupture in a shock wave tube, the characteristics method can be used to solve wave propagation equations. According to Fig. 1, Equation 1 is concluded [6].

$$1+\left(\frac{\acute{a}_0}{a_0}\right)\left(\frac{k_a-1}{k_h-1}\right)-\left[\left(\frac{\acute{a}_0}{a_0}\right)\left(\frac{k_a-1}{k_h-1}\right)\right]\left(\frac{P_e}{P_o}\right)^{\frac{k_h-1}{2k_h}}\left(\frac{P_o}{\acute{P}_o}\right)^{\frac{k_h-1}{2k_h}}-\left(\frac{P_e}{P_o}\right)^{\frac{k_a-1}{2k_a}}=0 \tag{1}$$

In which k_a and k_h are specific heat for air and helium, respectively, P_e is the middle pressure, P_O is the initial pressure of low pressure section, and $\acute{P}_O$ is the initial of the high pressure section, $\acute{a}_0$ is the speed of sound in the high pressure section and a_0 is the speed of sound in the low speed section.

3- Numerical calculation procedure

The 3D geometry that is used in this study for validation and simulation proposes is a cylinder with 55 mm diameter. The cylinder height is considered 20 and 8 mm, similar to the laboratory test conditions. Injector hole is considered to be a cylinder with 0.8 mm diameter and 1 mm length which is located in the center of the main cylinder. To reduce simulation computation time, the geometry is considered to be a half-cylinder shape due to its symmetrical shape.
To enhance calculation precision, finer meshes are used near the nozzle, its downstream and near wall-impinged plate region. The inlet boundary condition is assumed to be injector inlet fuel pressure. Also, side walls boundaries are assumed to be atmospheric pressure. The rest of the surfaces are considered to be simple walls.
After development and validation of the model, by using the available experimental data, methane injection will be simulated in a commercial engine called EF7.

4- Numerical data evaluation and simulation results

To check for a mesh independence solution, five different mesh sizes have been developed and simulated. In each case, the penetration length is calculated and compared with the others. Using the mentioned method, the optimum mesh number for 20 mm and 8 mm standoff distances are 410609 and 192975 meshes, respectively.
Transient helium gas injection into the atmospheric air is simulated using "k-ε" and "SST k-ω" turbulent models. In Fig. 2, comparison of radial penetration length for experimental and theoretical results are shown. Nominal injection pressure ratio is 3 and impinging wall distance is 20 mm.
To study the effect of the engine speed on injection jet performance, penetration depth is calculated for methane gas with 100 bar injection pressure, 40° crank angle BTDC injection start and four different engine speeds: 1000, 2000, 3000, and 4000 rpm. The results show that the engine speed

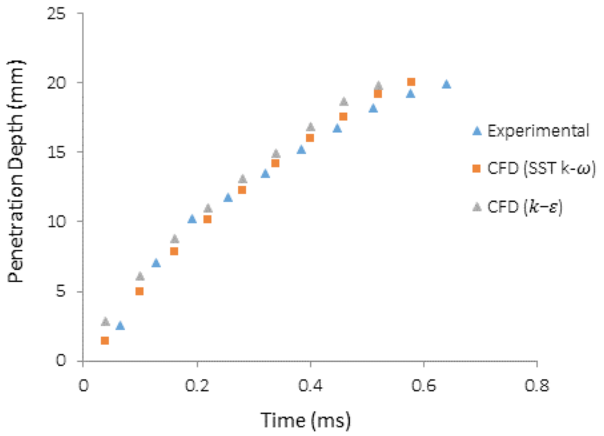

Figure 2. Comparison of the numerical and experimental data of axial jet penetration length before wall impinging with pressure ratio of 3

does not affect the penetration depth in the beginning of the injection, but at higher engine speeds the penetration depth is decreased. This result is in contrast with atmospheric injection.
To study the effect of injection timing, by assuming 100 bar inlet pressure and 3000 rpm engine speed and 1 msec injection duration, three different injection timing are studied: 40, 50, and 60° before top dead center. The results show that advancing or retarding the injection timing not necessarily leads to increasing or decreasing of the penetration depth. The penetration depth depends on other factors like inlet pressure or engine speed.

5- Conclusion

In this study, helium gas wall-impinging jet is simulated and validated by using experimental data. In the rest of the study, methane gas jet has simulated by a real engine conditions. The results of the study can be concluded as follows:

- The SST k-ω turbulent model predicts more precise penetration depth after wall-impinging compared to k-ω model.
- In atmospheric injection, moving the plate towards the jet increases penetration depth. But in closed chamber injection simulation, increasing engines speed leads to decreasing penetration depth.
- Increasing inlet pressure for each specific engine speed and injection timing ends in increasing penetration depth.

References

[1] zhao, F., D. L. Harrigton and M. la, "*Automative Spark-Ignited Direct Injection Gasoline Engines*", Warrendale: Society of Automobile Engineers Inc., USA, 2002.

[2] Chinto, F. and E. Tomita, "*Numerical Calculation of a Transient Methane Gas Jet Discharging into Quiescent Atmosphere at Mach One*", Graduate School of Natural Science and Technology, Dept. of Mechanical Engineering, 1991.

[3] Craft, T.J., L.J.W. Graham and B.E. Launder, "Impinging Jet Studies for Turbulence Model Assessment-II: An Examination of the Performance of Four Turbulence Models" *Int. J. Heat and Mass Transfer*, 36.10 (1993):

2685–2697.

[4] Afroz, F. and M. Sharif, "Numerical Study of Heat Transfer from an Isothermally Heated Flat Surface Due to Turbulent Twin Oblique Confined Slot-Jet Impingement", *International Journal of Thermal Sciences*, (2013).

[5] Ansys Fluent 14 theory guide, 2012.

[6] Shapiro, AH., "*The dynamics and thermodynamics of compressible fluid flow*", 1st ed. The Roland Press, 1953.

Please cite this article using:

A. Nassiri Toosi, A. HajiAliMohammadi, A. Heidary, "Numerical Study of a Wall-Impinging Gaseous Jet on a Flat Plate", *Amirkabir J. Mech. Eng.*, 49(1) (2017) 29-42.

DOI: 10.22060/mej.2016.729