Wang, Bo and Badawy, Tawfik and Hutchins, Peter and Tu, Powen and Xu, Hongming and Zhang, Xinyu (2017) Numerical investigation of the deposit effect on GDI injector nozzle flow. Energy Procedia, 105 . pp. 1671-1676. ISSN 1876-6102

Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 105 (2017) 1671 – 1676

Energy Procedia

The 8th International Conference on Applied Energy – ICAE2016

Numerical Investigation of the Deposit Effect on GDI Injector Nozzle Flow

Bo Wang[a,b], Tawfik Badawy[b], Peter Hutchins[b], Powen Tu[b], Hongming Xu[b,c,*], Xinyu Zhang[a]

[a]IDIC, University of Nottingham, 119 Taikang East Road, Ningbo 315100, China
[b]Furture Engine Research Group, University of Birmingham, Edgbaston, Birmingham B15 2TT, UK
[c]State Key Laboratory of Automotive Safety and Energy, Tsinghua University, 30 Shuangqing Rd, Beijing 100084, China

Abstract

Injector deposit is a common phenomenon for gasoline direct injection (GDI) engines that greatly affects the spray behavior and consequently the combustion performance and emissions. In this study, the deposit effect on the inner nozzle flow dynamics was numerically investigated. High resolution X-ray scan was performed first to obtain realizable information regarding to the nozzle and deposit morphologies and topology. Simulation was then carried out in the Large Eddy Simulation (LES) framework with cavitation taken into account by a homogeneous equilibrium model (HRM). It was found that the rough surface of deposit would lead to additional cavitation inception inside the counterbore and restrict the flow area, causing losses in the mass flow rate. Deposit inside the counterbore acted as an extension to the inner orifice and restricted the air recirculation, which was believed to be the cause of poor atomization of coked injector spray.

Peer-review under responsibility of the scientific committee of the 8th International Conference on Applied Energy.

Keywords: deposit, large eddy simulation, nozzle, cavitation, CFD

1. Introduction

Gasoline direct-injection (GDI) engines provide benefits such as high efficiency, low fuel consumption and low emissions; however, injector coking, otherwise known as the injector deposit effect, is a serious issue for them [1]. The injector coking can distort the carefully designed spray patterns and reduce the mass flow rate, which could lead to unstable combustion process and higher emissions [2–5]. Song et al. studied the deposit effect on the multi-hole GDI injectors spray behavior and claimed that the deposit would increase spray cone angle and reduce spray penetration [2]. Lindgren et al. studied the deposit effect on the swirl injectors with spray visualization technologies and found that the fouled injector

* Corresponding author. Tel.: +44(0)-121-4144153; fax: +44(0)-121-4143958.
E-mail address: h.m.xu@bham.ac.uk.

Peer-review under responsibility of the scientific committee of the 8th International Conference on Applied Energy.
doi:10.1016/j.egypro.2017.03.545

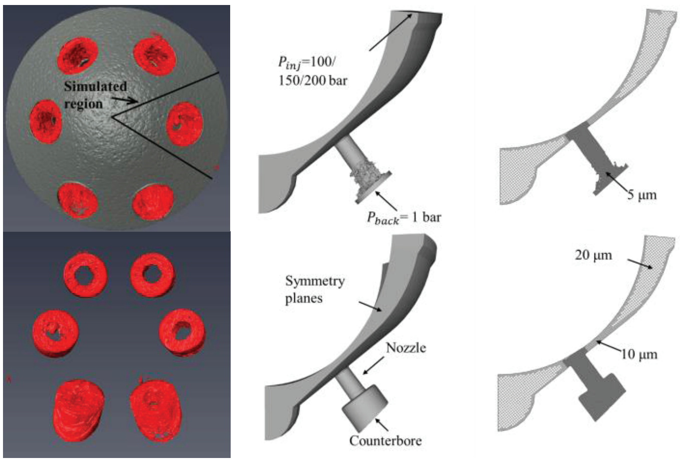
produced more dense and faster pre-jet compared to that of the clean one [3]. Joedicke et al. conducted an accelerated deposit formation test using additives that could accelerate the deposit formation [4]. After 55 hours dirty-up test, 23.5% fuel rate loss was observed accompanied with 20%, 93% increase of HC, CO emissions and 2.45% increase of fuel consumption. Wang et al. used two fouled and one clean multi-hole injector in a single cylinder spray guided DISI research engine to study the effect of injector coking on engine emissions [5]. They revealed that the fouled injectors consistently produced higher emissions. At the highest engine load of 8.5 bar IMEP, maximum difference was observed, where the fouled injector produced 58% higher PN emissions and 300% higher PM emissions.

In a recent work of Xu et al., a compensative review of injector deposit was conducted [6]. After reviewing the injector deposit formation mechanism, different methodologies used in injector deposit study, effect of injector deposit on engine performance as well as different strategies for injector deposit reduction, the author concluded that while extensive work had been done, the effect of GDI injector deposit was still not fully understood. Xu et al. pointed out that while CFD modelling can provide significant support in understanding the injector coking effect, limited work had been done as little information could be obtained of detailed structure and morphology of the deposits inside the nozzle [6].

Thus the aim of this study is to evaluate the deposit impact on the inner nozzle flow and cavitation process with the help of detailed 3D simulation. High resolution X-ray scan was performed first to obtain realizable information regarding to the nozzle geometry and the deposit morphologies.

2. Methodology

Clean and typical coked Bosch multi-hole injectors were used in this study. The coked injector was produced in the Future Engine Lab at University of Birmingham, having been used for 54 hours with Unleaded Gasoline (ULG95) under loads ranging from 3 to 8 bar IMEP, 150 bar injection pressure and 2000 rpm engine speed. Fig.1(a) shows the results of X-ray scan, where the red parts stand for the deposit. These images reveal that large amount of deposit had been formed, mainly inside the counterbore.



(a) (b) (c)

Fig.1. Schematic diagram of (a) scanned injector tip and deposit, (b) simulated regions and (c) numerical grids.

Simulations were carried out with the CFD software CONVERGE in the LES framework with a dynamic structure model [7] to account for the sub-grid stress. The HRM model developed by Schmidt et al. [8] was used to simulate the phase change between liquid and vapor (cavitation). Previous studies and validations with the same algorithm used on nozzle flow simulation can be found in Refs.[9,10].

A second-order central differencing scheme was used for the spatial discretization whenever was possible. An implicit Euler scheme was used for the time integration. Both a velocity-based Courant-Friedrichs-Lewy (CFL) below 0.2 and a speed-of-sound based CFL below 2.0 were used for the time step control. For computational cost consideration, only one injector hole was simulated at the maximum needle lift of 70 μm in this study, as shown in Fig.1. Constant pressure values were specified at both the outlet and inlet boundaries with a fixed back pressure of 1 bar and injection pressures of 100, 150 and 200 bar. Isooctane, as a typical surrogate for gasoline, was used as the fuel.

Numerical grids are shown in Fig.1(c). Both fixed embedding and adaptive mesh refinement methods were applied, with mesh sizes ranging from minimum 5 μm in the orifice and counterbore region to maximum 20 μm in the sac part. The total mesh number was around 900,000.

3. Results and discussion

3.1. Mass flow rate and model validation

In order to validate the CFD simulation model, the simulated mean mass flow rate (averaged in a time period equal to 15 times the time for a fluid particle to travel from the nozzle inlet to the counterbore exit [11]) was compared with the test data as shown in Fig.2(a). The experimental test was conducted with an in-house built long tube measuring system working in the Bosch method. Details of the test rig and experiment setup can be found in the Ref.[12].

As can be observed, the CFD results generally matched with the test data well and could clearly distinguish the difference between the clean and coked injector that the deposit caused around 9% mass flow rate loss. The deviations between the experimental and computational values were less than 4%. For the clean injector, CFD calculations slightly underestimated the experimental results for all three pressures, while for the coked injector, simulations slightly overestimated the results. This difference may be explained by the smoothing carried out on the deposit surface before used for the simulation. Due to the measurement uncertainties and the complexity of cavitating flow modeling, it was considered that the CFD code and setups were able to predict the flow behavior with a sufficient degree of confidence.

Fig.2(b) shows the grid independence analysis results under the condition of P_{inj}=150 bar and P_{back}=1 bar. Five different mesh sizes were tested ranging from 2.5 to 15 μm. It shows as the finest mesh size reached to 5 μm, the mass flow rate maintained stable, which demonstrated the CFD solution was relatively independent of the grid size. Therefore the finest mesh size of 5μm was used in this study as a compromise of accuracy and computational cost.

3.2. Effect of deposit on injector flow characteristic

Fig.3(a) presents the averaged the vapor volume fraction contours for the coked and clean injectors at both the mid-plane and four cross sections with P_{inj} = 150 bar and P_{back} = 1 bar. The mid-plane views indicated that cavitation only occurred at the orifice inlet for the clean injector. The sharp bend of the orifice inlet would lead to large velocity and pressure gradients for the flow, causing cavitation in this region. For the coked injector, however, except for the orifice inlet cavitation inception could also be found at the deposit surfaces inside the counterbore. The small projections and pits formed by the deposit caused flow separation and thus cavitation. These cavitation inception points can be more clearly viewed with the cross section views. The deposit caused cavitation developed along the deposit surface till the counterbore exit.

The velocity field is shown in Fig.3(b). Upstream as far as the counterbore, the velocity distribution appeared to be similar for the two cases. Inside the counterbore, clear air recirculation zones could be observed for the clean injector, which could promote the atomization of the spray. Nevertheless, for the coked injector, such air entertainment was constrained by the deposit which acted like an extension of inner hole, resulting in poorer plume dispersion and smaller spray cone angle. The absence of fuel/gas interaction was believed to be the cause of poor atomization of coked injector spray previously observed by other researchers such as in the Ref.[3]. A low velocity region could be observed at the upper surface of the counterbore deposit due to the existence of cavitation vapor, which further restricted the effective flow area. The presents of deposit and superior cavitation together restricted the effective flow area, resulting in the flow rate reduction shown in Fig.2.

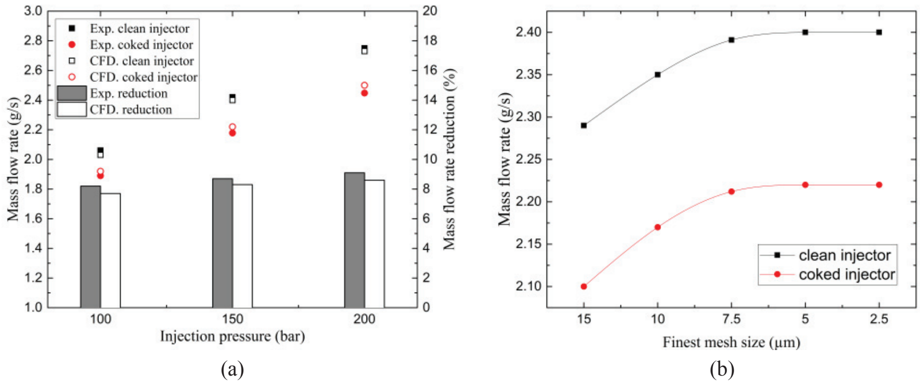


Fig.2. Comparison of mean mass flow rate from (a) experimental and numerical results and (b) different mesh sizes.

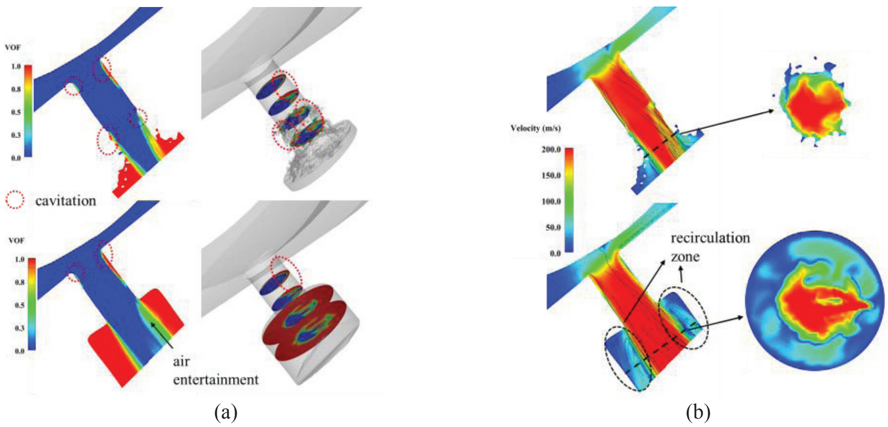
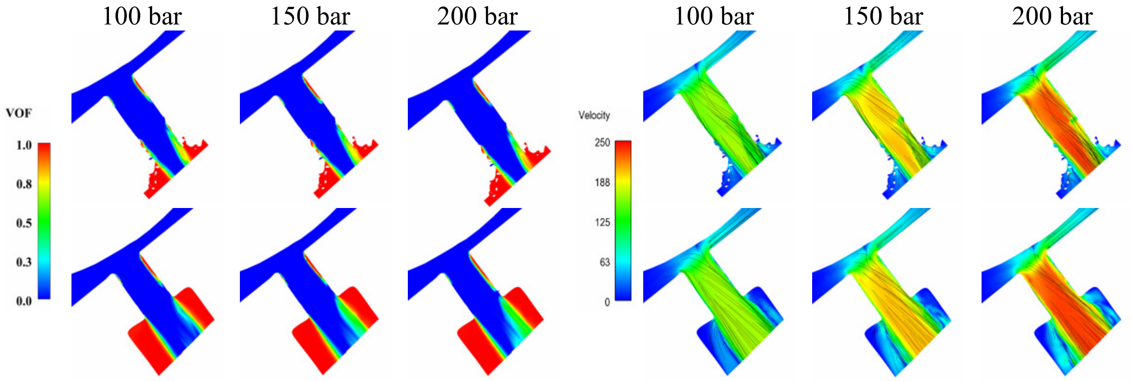


Fig.3. (a) Vapor fraction contours and (b) velocity contours for the coked and clean injector.

3.3. Effect of deposit under different injection pressure

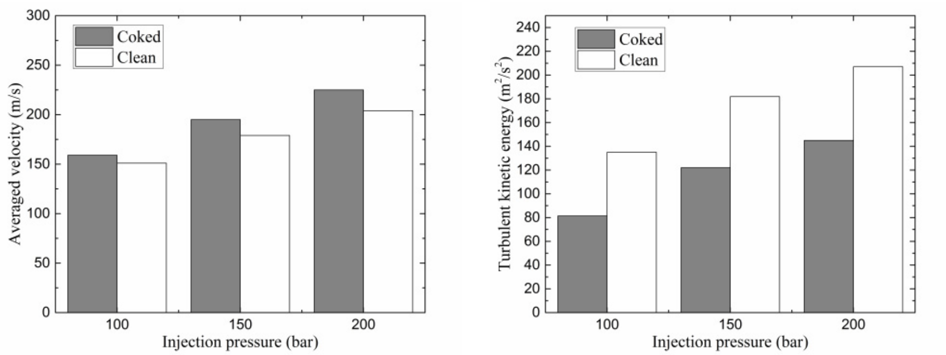
Fig.4 shows the CFD results under different injection pressures. The results demonstrated that the cavitation already occurred at 100 bar injection pressure. For the clean injector, the cavitation inceptions always occurred at the corners of orifice inlet. With higher injection pressure, the cavitation area expanded further downstream. For the coked injector, deposit caused additional cavitation inside the counterbore that could always be found at all three injection pressures. Higher cavitation level caused by the deposit leaded to further blockage at higher injection pressures, which resulted in the higher levels of mass flow rate loss observed in Fig.2.

Higher injection pressures also contributed to higher flow velocity and stronger fuel/air interaction, shown in Fig.4(b). More pronounced recirculation zones were found at the counterbore for the clean injector. Fig.5(a) presented a comparison of the mean exit velocities for the clean and coked injector. As a general trend, higher injection pressure leaded to higher exit velocities. Although the rough surface of deposit tended to increase the flow resistance, the restricted flow area and smaller cone angle of the flow resulted in the higher exit velocity. Apart from this, the absence of air/fuel interaction restricted the turbulence development, resulting in the much lower kinetic energies for the coked injector as shown in Fig.5(b). To be noticed, both the exit velocity and turbulent kinetic energy were calculated for the liquid flow region indicated by VOF$\leqq$0.5.



(a) (b)

Fig.4. (a) Vapor fraction contours and (b) velocity contours at different injection pressures.



(a) (b)

Fig.5. (a) Mean exit velocity and (b) mean turbulent kinetic energy of liquid flow at the counterbore exit.

4. Conclusions

In this work, the effect of deposit on the nozzle flow behavior was numerically investigated with detailed deposit morphology obtained from the X-ray scan. From the simulation results, following conclusions can be drawn:

- Deposit inside the counterbore acted as an extension of the orifice and restricted the air entertainment and recirculation, which was believed to be the cause of poor atomization of the coked injector spray.
- The rough surface of the deposit created additional cavitation inceptions inside the counterbore which restricted the effective flow area and resulted in the mass flow rate reduction.

- With higher injection pressure, cavitation area grew for both the clean and coked injectors. Deposit was observed to cause cavitation, leading to further blockage of the flow area.

Acknowledgements

The finical support by ESPRC Program at University of Birmingham with project code EP/N021746/1 is gratefully acknowledged. The authors also acknowledge the financial support from the International Doctoral Innovation Centre, Ningbo Education Bureau, Ningbo Science and Technology Bureau, China's MoST, University of Nottingham and the partial support by Ningbo Natural Science Foundation Program with project code 2013A610107.

References

[1] Zhao F, Lai MC, Harrington DL. Automotive spark-ignited direct-injection gasoline engines. Prog Energy Combust Sci 1999;25:437–562.

[2] Song H, Xiao J, Chen Y, Huang Z. The effects of deposits on spray behaviors of a gasoline direct injector. Fuel 2016;180:506–13.

[3] Lindgren R, Skogsberg M, Sandquist H, Denbratt I, Sandquist H. The Influence of Injector Deposits on Mixture Formation in a DISC SI Engine. JSAE 2003. d

[4] Joedicke A, Krueger-Venus J, Bohr P, Cracknell R, Doyle D. Understanding the Effect of DISI Injector Deposits on Vehicle Performance. SAE Tech. Pap., 2012.

[5] Wang C, Xu H, Herreros JM, Wang J, Cracknell R. Impact of fuel and injection system on particle emissions from a GDI engine. Appl Energy 2014;132:178–91.

[6] Xu H, Wang C, Ma X, Sarangi AK, Weall A, Krueger-Venus J. Fuel injector deposits in direct-injection spark-ignition engines. Prog Energy Combust Sci 2015;50:63–80.

[7] Pomraning E, Rutland CJ. Dynamic One-Equation Nonviscosity Large-Eddy Simulation Model. AIAA J 2002;40:689–701.

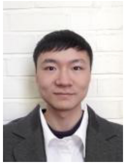
[8] Schmidt DP, Gopalakrishnan S, Jasak H. Multi-dimensional simulation of thermal non-equilibrium channel flow. Int J Multiph Flow 2010;36:284–92.

[9] Battistoni M, Xue Q, Som S. Large-Eddy Simulation (LES) of Spray Transients : Start and End of Injection Phenomena. Oil Gas Sci Technol - Rev d'IFP Energies Nouv 2016;71:24.

[10] Zhao H, Quan S, Dai M, Pomraning E, Senecal PK, Xue Q, et al. Validation of a Three-Dimensional Internal Nozzle Flow Model Including Automatic Mesh Generation and Cavitation Effects. J Eng Gas Turbines Power 2014;136:092603.

[11] Desantes JM, Salvador FJ, Carreres M, Martinez-Lopez J. Large-eddy simulation analysis of the influence of the needle lift on the cavitation in diesel injector nozzles. Proc Inst Mech Eng Part D J Automob Eng 2014;229:407–23.

[12] R.S Tay. Fuel Injection Rate Measurement Device. Univ Birmingham, Final Year Thesis 2012.



Biography

Bo Wang is a PhD student at University of Nottingham, China campus, with two years visiting research at University of Birmingham, UK. His research interests include both CFD simulation and optical diagnostics of fuel spray, two phase flow and combustion process in the internal combustion engines.