A study on a swirl type multi-hole fuel injector for a PFI system to improve air-fuel mixture formation

A J Alimin¹, Y KuangHe², A Sapit³ and H Selamat⁴
¹Head, Automotive Powertrain Research Group (ARG), Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia
²Bachelor Degree Project Student, Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia
³Principal Researcher, Flow Analysis, Simulation, and Turbulence Research Group (FAST), Faculty of Mechanical & Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Batu Pahat, Johor, Malaysia
⁴Director, Centre for Artificial Intelligence and Robotics (CAiRO), Universiti Teknologi Malaysia, 81310 Skudai, Johor, Malaysia
E-mail: ajais@uthm.edu.my

Abstract. This paper described the development and simulation works that were conducted on an injector that has a combined swirl spray and multi-hole injection functions. It was expected that those two functions would provide beneficial effects to the fuel injector, in order to provide better fuel and air mixing, especially for Port Fuel Injection (PFI) application. The combination of both functions was aimed for achieving high quality spray formation, which might be useful in cases when the combustion time is limited. Swirl spray pattern is known for fine droplets production with relatively smaller SMD and bigger coverage area in the combustion chamber. On the other hand, a multi-hole injector has the advantages of simplicity in manufacturing and operation. Preliminary results on the simulation works that were done separately, demonstrated the ability of the injector’s design to produce swirl spray as well as injecting near-symmetrical sprays from the four holes at the injector’s end. Nevertheless, these findings must be improved further in the future, especially on simulating the benefits of having a combined swirl and multi-hole functions inside a single injector. Furthermore, detailed assessment on manufacturability and cost effectiveness of this injector must also be addressed.

1. Introduction
According to BP 2016 Statistical Review of World Energy [2] energy consumption for emerging economies accounted for up to 97% of the increase in global energy consumption. Notable emerging economies countries [3] are forecasted to have increase access to transportation: either for commercials or personal usage [1]. As light duty vehicles such as motorcycles and compact cars are among the primary types of vehicles for personal transport, works on improving the fuel economy and reducing hazardous emissions for these types of engines and vehicles are always relevant and important.

A good air-fuel mixture is imperative for achieving efficient combustion for better fuel economy and emissions control, especially for a port fuel injection (PFI) system where the fuel delivery occurs a
distance away from the intake port. The quality of the mixture is highly influenced by the injection characteristics as well as the quality of mixing with the entrained air. For a PFI system, spray coverage and penetration are known to be the issues associated with conventional fuel injector. In this paper, the findings from a preliminary CFD work on the operation of a fuel injector with a swirl effect nozzle, which is designed to be used with small engines (capacity below 250cc), are presented, concerning the injector’s fuel delivery characteristics: spray and penetration.

2. Literature review
The conventional understanding of spray formation are described by Sazhin (2014) [6] to follow these stages: 1) development of a jet; 2) conversion of a jet into liquid sheets and ligaments; 3) disintegration of ligaments into relatively large droplets (primary break-up) and 4) break-up of large droplets into smaller ones (secondary break-up). The successful transformation achieved by the injected fuel, upon leaving the injector nozzle, in accordance to the abovementioned stages will ensure the achievement of an efficient combustion.

Prior to that, Mitroglou et al. (2006) [4] also emphasised the importance of also assessing the quality of spray structure, tip penetration and fuel droplets distribution, velocities and diameters as a function of nozzle design, as well as the value of the fuel delivery pressure. In their work, on the spray characteristic of a multi holes injector under different spray pressures, they found that the spray angle remains constant and is almost independent of injection and chamber pressures. In addition, they deduced that injector with a multi-hole and central hole will produce uneven flow distribution and consequently created over-penetration and unstable spray pattern.

For port fuel injection engines, mixture formation process starts with the injection of liquid fuel into the intake ports. The fuel droplets interact with the air inside the intake manifolds, as well as intake port walls and intake valves, creating fuel vapour and wall films. Therefore, injector design and intake port geometry are also influential to the fuel injection spray quality [5]. Their findings showed that, as the fuel pressure increased, both jet velocity and spray cone angle increased and as a result it makes the diameter of droplets much smaller and a better atomization was achieved. A better understanding of spray characteristics of port injector is also very important in matching spray parameters with different engine designs and targeted operations.

Recent researches, studies and experiments on the mixture formation and spray pattern are mostly done by using numerical analysis and computational fluid analysis as these methods are more detailed and provide more complete data for analysis. By carrying out the analysis using the correct operating boundaries and practical geometries, injection spray and mixture formation quality can be assessed and further recommended for actual physical experiment could be recommended. For the current work described in this paper, the proposed fuel injector will function using a central injection hole surrounded with swirl channels.

3. Methodology
The study was initiated with the design of the new injector nozzle, which has a central hole with swirl channels using CAD Software, Solidworks. The basic injector design was based on combination of a single path flow with swirl paths, which will converge into four outlets at the end of the injector. Then, using FLUENT software, the fluid flow simulations inside the injector were carried out. Based on this preliminary CFD works, the initial design will be improved and upon achieving an optimum performance through simulations, a prototype will be fabricated for future physical experiments.
Figure 1. The injector: plan view’s dimensions

Figure 2. The injector: side view’s dimensions

Figure 3. The multiple views of the reversed body of the injector representing the fluid inside

Figure 4. Incorporation of cylinder at the bottom of the injector

Figure 1 and figure 2 show the basic design of the swirl fuel injector and figure 3 depicted the reversed body of the injector representing the fluid inside it. Figure 4 shows the constructed cylinder in order to study the flow of the fuel after leaving the outlets, where the spray pattern can be observed. In this preliminary simulation, due to the difficulty in meshing, the simulations were divided into swirl-cylinder simulations and hole-cylinder simulations. The input and output boundary conditions were pressure-based, set at 3 MPa and 0 MPa respectively; the operating temperature was set at room temperature, and k-epsilon was the turbulent model used. Although they were made separately, it will form as the basis for future simulation works on the combined spray system. In both simulations, air and n-octane (as liquid and vapour forms) were used as the working fluids. For the swirl path, the initial angle was 105° and was then improved to 90°, to eliminate the formations of stagnation points inside the swirl paths.

4. Results and discussion
Preliminary simulation results are shown in figure 5 and figure 6, for the swirling behaviours inside the injector leading to the outlets. Whereas in figure 7 and 8, the simulation results after the fuel exiting the four outlets are shown, from bottom and side views.
High velocity flows were observed inside the swirl paths and the selected angle were able to eliminate stagnation points in between the swirl paths. The flows encircled the single injection path and moving downwards towards the four injector’s outlets. Nevertheless, the simulations were unable to simulate the combined flows from the swirl paths and the single injection path. High velocity streamlines were observed, as in figure 6, which could be due to the pressure difference between the inlet and outlet conditions, and they were too high in comparison to actual physical events. This observation will be subjected to further refinement in the future. In figure 7 and 8, the fuel flows exiting the four outlets were observed as expected. Even though the flows from the outlets are distinct and almost symmetrical, in accordance to the location of the outlets, the effects from having swirling paths inside the injector are not clearly demonstrated from these simulations.

The results obtained must be benchmarked in the future against simulations from the conventional multi-hole injector and also from swirl-only type injector. Those shortcomings will need to be investigated for future works for this type of fuel injector. Moreover, the positive effects of combining the swirl paths with the single injection must also be examined closely, in order to ascertain the influential factors for better fuel mixing inside the combustion chamber. Another aspect that must be
considered will be from the manufacturing perspective and the cost effectiveness of having this type of fuel injector as opposed to the conventional fuel injectors.

5. Summary
In this current work, a new configuration of fuel injector, which has a combined swirl spray and multi-hole functions had been successfully designed and investigated via simulations using CFD. Although the preliminary simulation works were carried out separately for the swirl paths flows and the single injection path flow, the results were useful to serve as the benchmarks for future works on this type of fuel injector. Swirling behaviour was observed and near-symmetrical flows were obtained at the outlets of the injector. It is recommended that for future works on this type of injector, the CFD simulations must attempt to combine and elucidate the benefits of having swirl spray function with the multi-hole injection. Furthermore, detailed assessment on manufacturability and cost effectiveness of this injector must also be addressed.

Acknowledgement
The authors would like to express their gratitude to researchers from Automotive Powertrain Research Group (ARG) and Flow Analysis, Simulation, and Turbulence Research Group (FAST), for the endless valuable support in conducting this work. The financial support from APT TOUCH SDN BHD is also acknowledged.

References
[1] Asif M and Muneer T 2007 Renewable and Sustainable Energy Reviews, 11
[2] Dudley B 2016 BP Statistical Review
[3] Hoskisson R E, Eden L, Chung M L and Wright M 2000 The Academy of Management Journal, 43 3
[4] Mitroglou N, Nouri J M, Gavaises M and Acroumanis, C 2006 International Journal of Engine Research 7 3
[5] Movahednejad E, Ommi F and Nekofar K 2013 EPJ Web of Conferences, 45
[6] Sazhin S 2014 Droplets and Sprays (London: Springer)