Proceedings of the
International Conference on Mechanical Engineering2011
(ICME2011) 18-20 December 2011,Dhaka, Bangladesh

ICME 11-FL-032

EFFECTS OF GUIDE VANE SWIRL AND TUMBLE DEVICE (GVSTD) TO THE AIR FLOW OF NATURALLY ASPIRATED CI ENGINE

Idris Saad[1, 2] and **SaifulBari**[1]

[1]Barbara Hardy Institute, School of Advanced Manufacturing and Mechanical Engineering, University of South Australia, Australia
[2]Fakulti KejuruteraanMekanikal, UniversitiTeknologi MARA Malaysia

ABSTRACT

Current trend shows that biodiesel has a high potential as an alternative fuel for internal combustion engines to support depleting petroleum based fuel resources. However, the viscosity of biodiesel is heavier as compared to diesel fuel and gives a problem of low atomization, resulting in lower performance of the engine. In this paper, a set of vanes, named Guide Vane Swirl and Tumble Device (GVSTD), are added into the air intake system to create controlled turbulence to improve the atomization of biodiesel. To observe these phenomena, a simplified in-cylinder simulation model consists of fluids, solid parts,limited to the intake,compression and part of expansion strokes were drawn using SolidWorks. The model was then exported to ANSYS-CFX software where three techniques, immersed solid, moving mesh and multiple analyses, were combined to simulate the model and visualize the characteristics of the fluid flow with GVSTD. The heights of the vanes were also varied to optimize the effect of turbulence generated by GVSTD. The results of velocity, turbulence kinetic energy (TKE) and pressure are compared to justify the significance of this proposed modification.

Keywords: Guide Vane Swirl and Tumble Device (GVSTD), Compression Ignition Engine, Turbulence, In-Cylinder Air Flow.

1. INTRODUCTION

Petroleum based fuel that has been used as one of the primary sources of energy for transportation, since the early birth of automobilesis reportedly being depleted. This news is enough to push the researchers searching for other sources of energy for internal combustion engines (ICEs). Currently,the world has a huge number ofautomobiles on the road using ICE. Therefore,the changes cannot be drastic to avoid shock to the automotive industries, markets, users and others.Biodiesel can be directly used in an existing diesel engine with minor modifications [1, 2].Hence, biodiesel can be a potential alternative fuel to replace or supplement depleted crude oil [3].

Biodiesel is renewable.This means that the CO_2 produced by combustion can be consumed by the plants producing biodiesel seeds, so the net emission of CO_2 from biodiesel combustion is zero. The emission of toxic gases like CO, HC, aldehydes, particles and black smokes are also reported to be lower, in comparison to burning diesel fuel, except $NO_{X,}$ which is higher[1-4].

Beside the advantages mentionedabove, the engine performance and fuel consumption are slightly poorer than conventional diesel fuel[4].The density and viscosity of biodiesel are found to be higher than ordinary diesel fuel [5, 6]. Furthermore, biodiesel has heavier molecules than diesel, which tends to remain unburned and deposit inside the combustion chamber.

To overcome these, one of the solutions is to make the in-cylinder air flow much more organized, to break these heavy molecules and mix them with air during compression/combustion stroke. This conceptoforganized in-cylinder air flow was implemented with various techniques for petrol and diesel engines[7-16]. However, todate none of the reports was found to design and testspecifically for engines using biodiesel. The idea of more organized air flow is to break-up the higher viscous and heavier fuel molecules of biodiesel and mix them properly with air. This will eventually improve the combustion and reduce the carbon deposits inside the combustion chamber.

This concept of organized air flow is not new and was firstfound for carbureted engine using gasoline by using rotating blades placed between carburetor and intake manifold to provide more air swirl for efficient air-fuel mixing[7]. A setof the bladeswith anglesvarying from $2^{\circ}$to 45°[8] was stated to improve the performance of the engine since early development of carbureted engine faced the problem of air-fuel mixing. Subsequently, fuel molecule breaker or atomizerwas patented with various names, shapes, sizes, designs and engine types[9-16], which were successful to improve the engine performance.

Yet the main objective of those designs was to generate controlled air turbulence to mix air and fuel homogenously and increase the combustion efficiency.In this research, controlled air flow is sought to break the higher viscous and heavier molecules of biodiesel, and then mix them with air for better combustion and lower carbon soot. Thus, this paper presents the results ofsimulation of airflow in a diesel engine with and without Guide Vane Swirl and Tumble Device (GVSTD),to understand the air flow characteristics and its effects to the air flow of a naturally aspirated (NA)diesel engine, before the GVSTD is designed and tested for biodiesel in diesel engines.

2. METHODOLOGY

Computer simulation of a diesel engine was used to visualize the characteristics of in-cylinder air flow. This method includes drawing the model in SolidWorks 2010and CFD analysis was carried out with ANSYS CFX 12.1.

2.1 Drawing of the Models

The model of the engine was drawn with SolidWorks according to the drawing of HINO 40WD diesel engine coupled to a generator. The basic specifications of the engine are listed in Table 1.

Table 1: Basic specification of the engine simulation model

Bore × Stroke	: 104 × 108 mm
Number of cylinders	: 1
Compression Ratio	: 17.33
Engine speed	: 1500 rpm

The drawing onlyconsidered the important geometrical aspectof the airflow of the engine such as, intake runner, intake valve and cylinder. Cylinder and intake runner were assigned as fluid domains, while intake valve was assignedas solid domain.The base model withthe cylinder in full is shown in Fig. 1. This base model without GVSTD was used to generate reference data to compare the results with any design modifications.

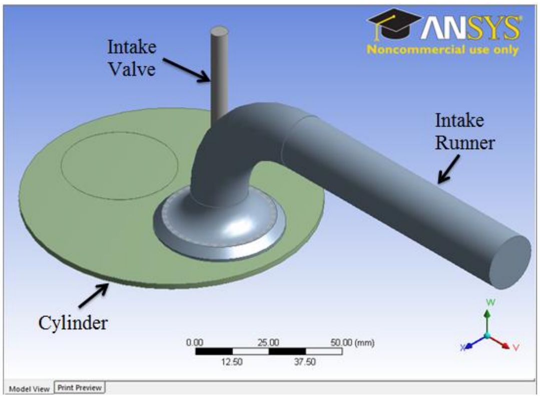

Fig 1. Drawing model from isometric view of base model and its parts

In order to develop GVSTD, few considerations were taken. It has been shown by researchers that more blades inside the intake system will increase the resistance of air flow to the cylinder and reduce the volumetric efficiency [9, 11-13, 15]. However, simple models developed by Kim [9, 10] also providedmoderate satisfactorypositive effects to the engine. Considering these factors, four blades with 30 mm in length were selected. The blade angles werealso varied by researchers from 2° to 60° [16]. Considering their results, the blade twist angle of 35° was chosento be suitable for this research.

Another objective of this project is to investigate the effect of the height of the blades to the in-cylinder air flow.Therefore, the heights of blades were variedat 25%, 50% and 75% of intake runner radius,designated as 0.25R, 0.50R and 0.75R where R is the radius of intake runner. The basic specifications of GVSTD is tabulated in Table 2 and shown in Fig. 2.

Table 2: Basic specification of the GVSTD

Number of Blades	: 4
Blades Twist Angle	: 35° Clockwise direction
Blades Length	: 30 mm
Radius of Intake Runner (R)	: 10 mm
Height of Blades	:0.25R = 0.25 × 10mm = 2.5mm : 0.50R = 0.50 × 10mm = 5.0mm : 0.75R = 0.75 × 10mm = 7.5mm

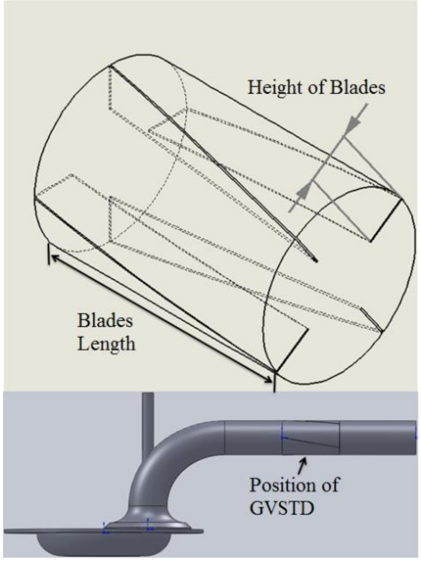

Fig 2. Basic configuration of GVSTD and its position in the model

2.2 Simulation Setting

The CFD simulation setting for the model was prepared according to the standard internal combustion engine simulation setting of ANSYS CFX [17]. It is includedin the initial domains setting of the two transient analyses of intake and compression strokesas explained below.

The fluid domains of the models were set as air and the air was considered as ideal gas for CFD setting. The setting also included the heat transfer and turbulence model as total energy and shear stress transport (SST) respectively. Moreover, the model was set to draw air from the atmosphere at 0 m/s velocity and 300 K temperature. But, for the intake valve, it is solid in nature and must move according to the valve timing of the actual engine. For this reason, immersed solid was used to fulfillthe requirements. The initial settings of the simulation were set approximately, according to real conditions of the engine to achieve realistic results [17, 18].

In order to run CFD simulation, separate analyses wereprepared, based on the sequence of intake and compression/expansionstrokes. Theintake stroke involves the piston movement from top dead centre(TDC) to bottom dead centre (BDC) and creates a negative pressure to draw the air from atmosphere. As a result,the intake CFD analysis required three components: cylinder, intake runner and intake valve, to complete theirfunctions. But for CFD analysis of compression, only cylinder domain wasconsidered as shown in Fig. 3.This is because in real application, the compression stroke occurs ina closed system with closed intake and exhaust valves, without any effectsfrom intake and exhaust runners. However, the CFD simulation for compression strokerequired the end results of intake analysis and therefore, a link between these two analyses were generated.

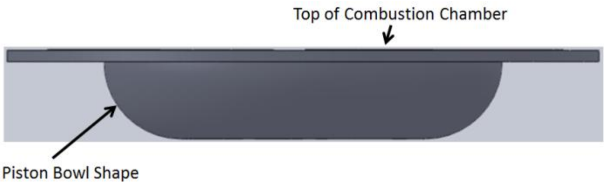

Fig 3. Side view of cylinder domain

3. RESULTS AND DISCUSSION

This section presents the analyses of velocity, turbulent kinetic energy (TKE) and pressure inside the combustion chamber from 340° to 370° crank angle (CA).The chosen parameters are the key factors of in-cylinder air motion.ThisCA range was selected to cover the periods before the start of injection (SOI) to the end [19]. Therefore, it will envisage the first part of premixed combustion phase to predict early development of combustion.Also, this range of CA willdetermine the influenceof mixing on the combustion propagation and determine its efficiency. Thus, it is important to prepare effective in-cylinder air motion in this range.

3.1 Velocity

In-cylinder air velocity plays an important role to distribute the injected fuel to mix with the air [19, 20].High in-cylinder velocity will give an advantage to the higherviscous biodiesel to spread and mix with the air before combustion occurs. Furthermore, it can improve the flame propagation when combustion commences and reduce the time to burn[11]the whole injected fuel. Burning the whole fuel means increased combustion efficiency and produces more power. Additionally, it can reduce the amount of the un-burnt hydrocarbon and may help reducing harmful exhaust products [2, 11, 20].

In reference to Fig. 4, all GVSTD models show the same pattern of in-cylinder average velocity profiles. The highest velocity of air inside the cylinder is achieved at crank angles ranging from 340° to 355° which are important for the injected fuel to break up and mix with air. The main objective of this research is to break up higher viscous biodiesel. Therefore, 0.25R GVSTD vanes which achieved the highest velocity of around 4.8 m/s would be better than other GVSTDs.

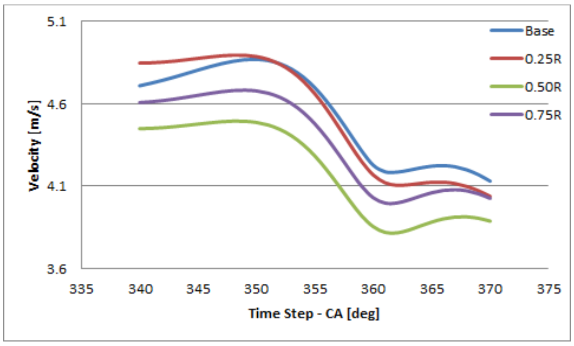

Fig 4. Average velocity profile in-cylinder

Following this, the graph shows a reduction of velocities until 360°. This reduction occursas the piston is decelerating toward TDC and at TDC the piston velocity is zero.Also, since cylinder volume decreases, the pressure increases. The increase of in-cylinder pressure will keepthe air molecule close and increase collisions among them.Hence, it will limit the movement of air and produce low velocity.

However, after TDC the initial momentum created by the intake runner with GVSTD is gone and the in-cylinder air-motion is only created by the descending piston in the expansion stroke. Again, the 0.25R GVSTD shows a velocity of 4 m/s which is higher than other GVSTDs. However, with combustion in real-cases this velocity will be much higher.

In conclusions, comparing the graphs between base model with and without GVSTD (Fig. 4) shows that at the first section of the graph (before 350°) 0.25R GVSTD shows the highest velocity. This result is expected on the basisof the successful pattern mentioned in the introduction. Additionally, by using GVSTD, the velocity distribution is more evenly distributed, as shown in Fig. 5 (e), (i) and (m) with larger greenthan red coloras compared to Fig 5 (a)i.e., without GVSTD. This results is also established by Elkotb, M.M., et al.,[18] when velocity vector is plotted for different configurations of inlet manifold geometry and Reeves, M., et al.,[21]for the barrel swirl breakdown. Since 0.25R have the lowest height of vane in comparison to other GVSTD models, it will reduce the possibility of GVSTD to become an obstacle to the flow as explained in section 2.1 above.

However, Fig. 4 also shows that the base model shows higher velocity after TDC.In order to understand this result, Fig 5(b) is referred.Thefigure plots the velocity contour at 1 mm from the top of the combustion chamber wall and denoted as plane 1.On that figure, it is shown that higher velocity accumulated at the centreofthe cylinder in the shape of a ring.

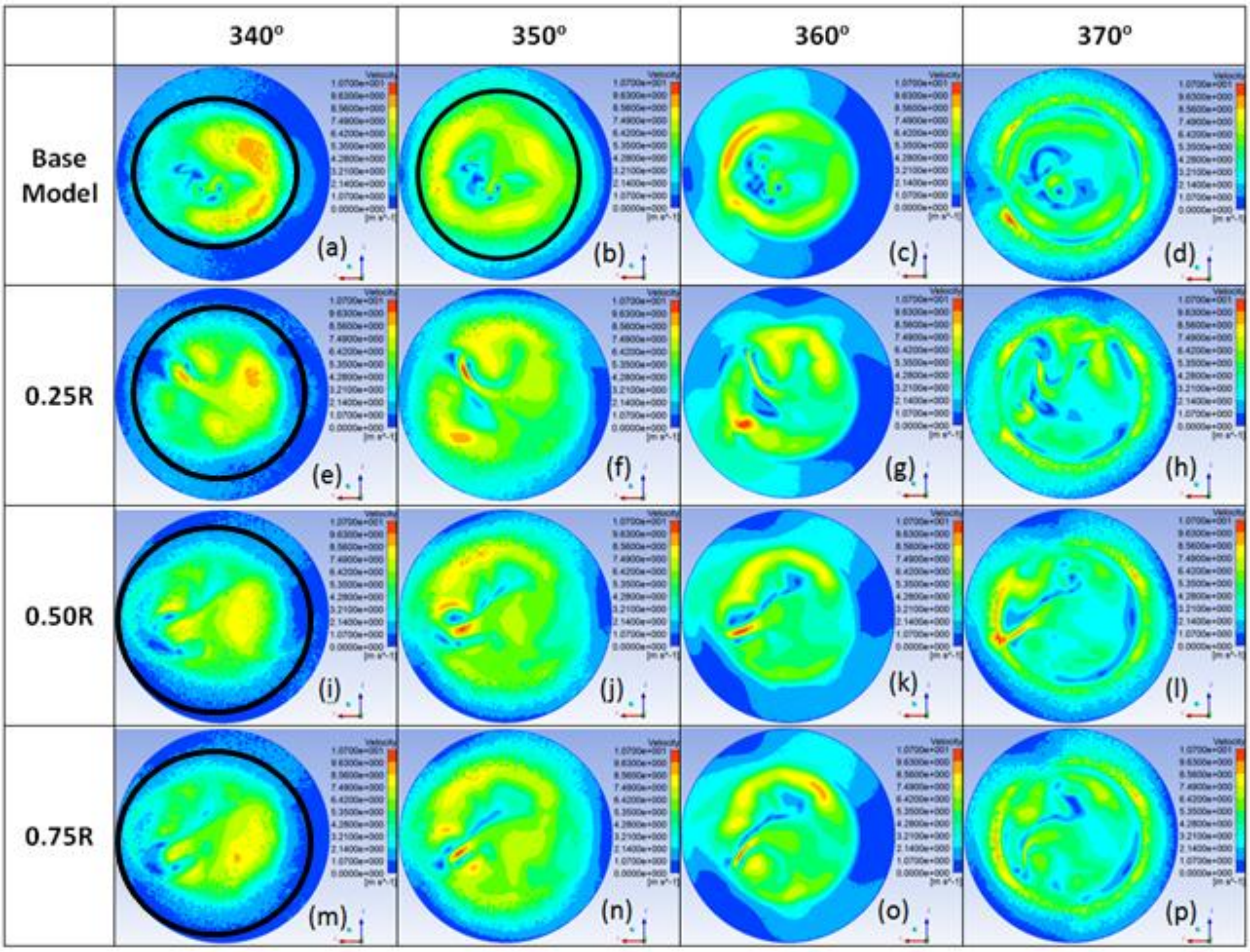

Fig 5. Top view velocity contour at plane 1 (1 mm to the top of combustion chamber wall)

In addition, whilst observing the geometry of cylinder in Fig. 3, it is found that the minimum space is at the edge of cylinder while the piston bowl is placed at the centre. Combining these two observations, higher average velocity at the centre of the cylinder with higher space allows maximum average velocity to occur as mention in section 3.But for GVSTD as explained above and proved by Elkotb, M.M., et al.,[18] and Reeves, M., et al.,[21], the velocity distribution occupying the entire cylinder will give lower average velocity as compared to the base model.

Since plane 1 is close to the position of fuel injector, it is important to have higher velocity here. This is to ensure injected fuel can have sufficient air-velocity to breakup and mix with available air during the short period of time. It became more significant at plane 1, where injected fuel is initially received.

Therefore, having higher velocity within the range of injection at plane 1, gives an advantage to heavier molecules of biodiesel to disperse moreand mix with the in-cylinder air.Based on that, Fig 6 illustrates the curve of maximum velocity at CA for all models. It shows that 0.25R is far better than the base model, with the increase of approximately 22.5%.This result is the key factor to claim that with the right configuration of GVSTD, it will improve the in-cylinder air flow. Unquestionably, this higher velocity will assist in spreading the higher viscosity biodieselfurther, resulting in better combustion and lower carbon deposits.

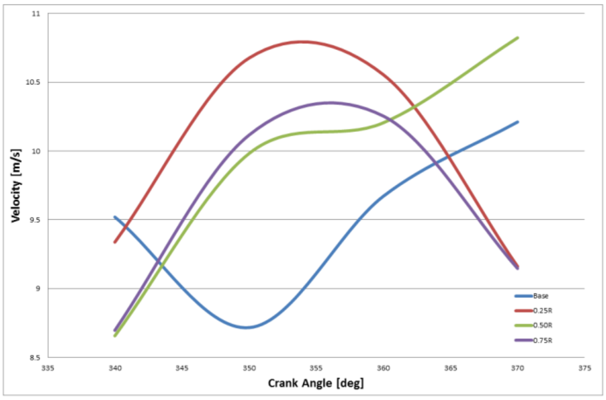

Fig 6. Maximum velocity at plane 1

3.2 Turbulent Kinetic Energy (TKE)

The behavior of the average TKE shown in Fig 7 is a result of average velocity in Fig 4, where TKE is the cumulative eddies of its kinetic energy.It shows that at the first part of the graph, the TKE is higherof the base model than other models. This is because of the large accumulated velocity of base model shown in Fig 5 (b) gives a good effect to the TKE, particularly around SOI to TDC. This is vital to receive the injected fuel, breakup and mix with the air during the ignition delay period. However, after that, because 0.25R has the highest maximum velocity as shownpreviously (Fig 6), its eddies can also maintain until the beginning of the expansion stroke which can lead to better flamepropagation and better combustion. Therefore 0.25R give more advantage than other models since TKE is used to spread the flame in mixing-control combustion regions rather than spreading the fuel at the early stage of injection.

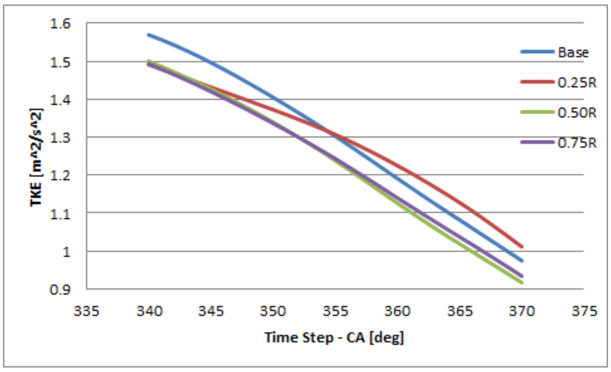

Fig 7. Average TKE profile in cylinder

3.3 Absolute Pressure

Fig 8 demonstrates the linear relationship between average absolute pressure and the height of the blades from 340° to 370° and its value at 360° is tabulated in Table 3. It means increased height of the blades will give greater pressure.

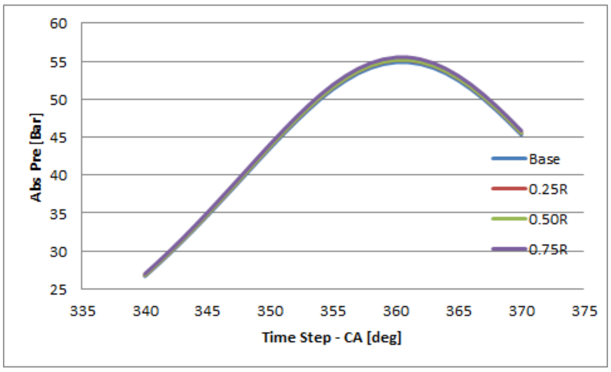

Fig 8. Average absolute pressure profile in cylinder

Table 3: Absolute Pressure at 360°[bar]

Base Model	0.25R	0.50R	0.75R
53.842	54.126	54.15	54.492

The increase of in-cylinder pressure is less at the commencement of compression due to the inlet valve staying open.Without GVSTD, the random air flow easily returns to the intake runner, while the piston is moving towards TDC. With GVSTD, it can be imagined that the flow goes into the cylinder during intake stroke in an organized clockwise direction like tightening a screw.The breakup charge on top of the piston (when the piston is moved towards TDC) is difficult to change direction, since more flows are coming in, in a clockwise direction. Due tothiscircumstance, the air is trapped in the cylinder and the amount of air is increased.This claim is confirmed by raising the pressure in the same cylinder volume and boundary settings.Therefore, another advantage of using GVSTD is found.

Higher viscosity of biodiesel has caused higher penetration length and lower spray angle resulted more carbon deposits on the piston head and less air utilization. The terminologies are illustrated in Fig. 9.It is expected that high in-cylinder pressure will reduce the penetration of the injected fuel and accumulate in the center of cylinder underneath the injector [19]. Because of this fact, the combustion will occur nearby TDC, just outside the surface of the mist. The round shape of the initial flame will expand evenly with the capability of TKE and velocity to propagate the flame and produce smooth combustion, while the injector continues injecting the fuel and gives a better combustion efficiency to reduce the existing difficulties of engine running on biodiesel.

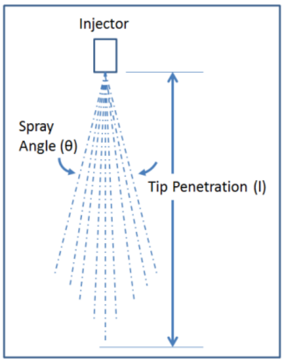

Fig 9: Schematic fuel spray from diesel injector

4. CONCLUSIONS AND RECOMMENDATIONS

Based on the explanation of the results and discussion above, it is conclusivethat GVSTD can provide the positive effect to the in-cylinder air flow for a biodiesel engine.Different heights of the blades will give different results, either in positive or negative ways.This paper acknowledges that 0.25R give better results compared to others asit gives highest velocity at SOI near the injector and highest average TKE. It can also increase the pressure better than the base model to reduce the penetration of injected fuel and have a potential to make the spray angle wider than the base model.

Greatersurface area of swirl device,supported by Reeves, M., et al., [21], gives better flow in terms of strong circulation of its velocities in support of this paper. Based on the presented results, the concerns of GVSTD becoming an obstacle to the flow is eliminated. Rather it gives an advantage to hold the air during the initial compression to increase the amount of the air.This advantage will reduce the core of injected fuel and expand the fuel spray angle.

High in-cylinder pressure can additionally hold the fuel at the centre of the cylinder and avoid them contacting the cylinder wall.This is because the cylinder wall is the coolest position, where it is surrounded by a water jacket. Hence, when fuel contacts this surface, it reduces the possibility of fuel to burn. Furthermore, the cylinder wall is also covered by a layer of lubricant oil. Therefore, combustion at the cylinder wall will increase carbon deposit resulting in black smoke and aldehydes.

Moreover, this paper suggests that further analysis of GVSTD, in terms of its parameters and design, must be conducted in order to achieve the optimized design.The simulation also needs to include the injection process and follow up ofthe experiment,prior tothe certain claim that GVSTD will improve the performance of biodiesel engine.

5. ACKNOWLEDGEMENTS

The first author would like to acknowledge the support by UniversitiTeknologi MARA and Ministry of Higher Education, Malaysia for the scholarship granted to carry out this research.

6. REFERENCES

1. Murugesan, A., et al., *o as an alternative fuel for diesel engines--A review.* Renewable and Sustainable Energy Reviews, 2009. **13**(3): p. 653-662.
2. Ramadhas, A.S., C. Muraleedharan, and S. Jayaraj, *Performance and emission evaluation of a diesel engine fueled with methyl esters of rubber seed oil.* Renewable Energy, 2005. **30**(12): p. 1789-1800.
3. Demirbas, A., *Importance of biodiesel as transportation fuel.* Energy Policy, 2007. **35**(9): p. 4661-4670.
4. Carraretto, C., et al., *Biodiesel as alternative fuel: Experimental analysis and energetic evaluations.* Energy, 2004. **29**(12-15): p. 2195-2211.
5. Kalam, M.A. and H.H. Masjuki, *Biodiesel from palmoil--an analysis of its properties and potential.* Biomass and Bioenergy, 2002. **23**(6): p. 471-479.
6. Benjumea, P., J. Agudelo, and A. Agudelo, *Basic properties of palm oil biodiesel-diesel blends.* Fuel, 2008. **87**(10-11): p. 2069-2075.
7. Horscroft, T., *Auxiliary Air Inlet*, in *United States Patent Office*1932: USA. p. 4.
8. Charles W. Thomas, J., *Fuel/Air Mixing Device for Internal Combustion Engine Carburator*, in *United States Patent*1976: USA.
9. Kim, J.S., *Fluid Swirling Device for an Internal Combustion Engine*, in *United States Patent*2003: USA.
10. Kim, J.S., *Fluid Swirling Device*, in *United States Patent*2006: USA.
11. Shyh-Shyan Lin and J.-C. Yang, *Intake Swirl Enhancing Structure for Internal Combustion Engine*, in *United States Patent*2000: USA.
12. Cheng, T.Y., *Gas Swirling Device for Internal Combustion Engine*, in *United States Patent*2003: USA.
13. Zetmeir, K.D., *Self Contained Air Flow and Ionization Method, Apparatus and Design for Internal Combustion Engines*, in *United States Patent*2002: USA.
14. Md Iqbal Mahmud, H.M. Cho, and K. Sang-Shin, *Variable Countercurrent Distribution Control (VCDC) System in IC Diesel Engine.* Proceeding of the World Congress on Engineering, 2009. **II**.
15. Kim, J.S., *Air Turbulence Generator of Internal Combustion Engines*, in *United States Patent*2000: USA.
16. Lyssy, N.G., *Fixed Blade Turbulence Generator*, in *United States Patent*1982: USA.
17. ANSYS, *Chapter 32. Internal Combustion Engine*, R. 12.0, Editor 2009.
18. Elkotb, M.M., et al., *Effect of the Inlet Manifold Geometry on Swirl Intensity at the End of Compression Stroke for Open Combustion Chamber Diesel Engine.* The Fourth International Symposium COMODIA 98, 1998.
19. Heywood, J.B., *Internal Combustion Engines Fundamentals*1988: McGraw Hill International.
20. Benajes, J., et al., *The effect of swirl on combustion and exhaust emissions in heavy-duty diesel engines.* Proceedings of the Institution of Mechanical Engineers -- Part D -- Journal of Automobile Engineering, 2004. **218**(10): p. 1141-1148.
21. Reeves, M., et al., *Barrel swirl breakdown in spark-ignition engines: insights from particle image velocimetry measurements.* Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering, 1999. **213**(6): p. 595-609.

9.MAILING ADDRESS

Idris Saad
Barbara Hardy Institute, School of Advanced Manufacturing and Mechanical Engineering, University of South Australia,
Mawson Lakes Campus, SA 5095, Australia