UNIVERSITI TEKNOLOGI MARA

PROSIDING



Seminar INSIGHTS 2002
Universiti Teknologi MARA

BRC (Biro Penyelidikan & Perundingan), Universiti Teknologi MARA, 40450 Shah Alam

Biro Penyelidikan dan Perundingan
Universiti Teknologi MARA
40450 Shah Alam, Selangor
Malaysia

© 2001. Biro Penyelidikan dan Perundingan. Hak cipta Terpelihara.

Sila majukan sebarang pertanyaan kepada:

Penolong Naib Canselor(Penyelidikan)
Biro Penyelidikan dan Perundingan
Universiti Teknologi MARA
40450 Shah Alam
Selangor, Malaysia

Tel : 603 5544 2094 / 2093 / 2092
Fax : 603 5544 2096
e-mel : azniz132@salam.uitm.edu.my



Jawatankuasa Penyunting

Penaung	:	Y. Berbahagia Datuk Prof. Dr. Ibrahim Abu Shah Naib Canselor Universiti Teknologi MARA
Penasihat	:	Yang Berbahagia Dato' Prof. Dr. Abdul Halim Mohd.Nawawi Timbalan Naib Canselor (Pembangunan dan Penyelidikan Universiti Teknologi MARA
Ketua Penyunting	:	Prof. Dr. Azni Zain Ahmed Penolong Naib Canselor(Penyelidikan) Universiti Teknologi MARA
Penyunting	:	Prof. Madya Dr. Zainon Mohd. Noor Koordinator Penyelidikan (Sains dan Teknologi) Prof. Madya Dr. Rosmimah Mohd.Noor Kooordinator Penyelidikan (Sains Kemasyarakatan dan Kemanusiaan)
Lembaga Penyunting:		Prof. Madya Ir. Dr. Hj. Abdul Rahman Omar Dr. Rashidah Abd. Rahman Cik Dapeah Ahmad

NEW FUELLING STRATEGIES FOR PORT INJECTED SPARK IGNITION ENGINE TOWARDS IMPROVEMENT IN FUEL CONSUMPTION AND EXHAUST EMISSION

Hakim A Abas
Saqaff Alkaff
Abdul Rahman Omar
C S Ow
Fakulti Kejuruteraan Mekanikal

ABSTRACT

Multi-point fuel injection (port injection) became popular with most passenger cars as it eliminated most fuel transportation problems through the induction system hence providing better fuel distribution, fuel consumption and less exhaust emission. However, fuel deposition on the inlet port wall and on the valve stem and shoulder, in particular, during transient and low speed running forced many car manufacturers to investigate other fuel injection techniques as alternatives.

This paper reports on an effort towards improving port injection performance employing two and three-dimensional computer modelling studies for air flow and fuel spray behaviour for port injected engine. Further Computational Fluid Dynamics (CFD) simulations employing FLUENT software were carried out for direct injection based on the internal dimensions of a WIRA 1.6 L engine.

A Laser Imaging System is used to investigate the fuel spray development in terms of the spray angle, spray penetration and the radial axis. The fuel spray characteristics in terms of the two components of fuel droplet velocities and the fuel droplet diameters were studied using the Particle Dynamic Analyser (PDA) system. Fuel spray generated from a fuel injector was used in the experimental investigation. The fuel injector used was from the fuelling system of a WIRA 1.6L engine with modifications for an extension. It was found that the spray angle stays at 36.8 degrees at all engine speeds. This is because the fuel pressure at the nozzle inlet (at the main trial) is constant. Results reveal that droplet diameter is in the range of 65 microns to 80 microns. The diameter of droplets also showed a tendency to increase with higher engine speed.

Another specific objective of the project is to employ Computational Fluid Dynamic (CFD) simulations in the study of fuel transportation and mixing behaviour in port-injected engines. This was subsequently extended to simulations of direct injection into the combustion chamber. Various designs of a modified combustion chamber and a suitable piston head shape to suit direct injection operations are reported. A study, at different engine speeds, of spray locations and fuel injection angles were conducted. The maximum air velocity was found to be at the exit of the air inlet valve. Two

highly turbulent domains were observed on both sides on the inlet valve and a stagnation domain below the valve face note. The spray angle has to be off-setted by about 30 degrees relative to the valve stem axis in order to give significant spray distribution in the cylinder.

INTRODUCTION

The operating characteristics of spark-ignition (SI) engines depend strongly upon the physical state of the air-fuel mixture induced into the cylinder. The induction system plays an important role in air-fuel mixture preparation, which gives some kind of ratio required for the engine needs. This consists of fuelling delivery system and the manifold. Different arrangement of manifold and cylinder provide quite different air and fuel distribution problems.

There are several types of induction systems which differ from each other by the way fuel is introduced and which affects the composition of the air-fuel mixture. The common fuelling systems are the carburetted system, single-point fuel-injection system with one or two electronically controlled injector and port injection. Typical problems inherited with carburetted and single-point fuel injected engines such as transportation, fuel mal-distribution, transient air-fuel ratio excursions and cyclic combustion variation could be eliminated by the implementation of port-injected (multi-point) fuelling systems. These enables the fuel to be introduced closed to the inlet valves and hence reduces the problems associated with the fuel transportation through the induction and generally better fuel control strategy can be achieved compared with carburetted and single point injection system. However, port-injection has its own problems that can severely affect engine performance. A major problem which is inherent with the port-injected-fuel-system is the fuel deposition on the inlet port wall after the fuel is injected and as the inlet valve starts to open. This problem becomes more complicated during slow running (town drive), as more fuel will impinge on the wall due to the low air stream velocity.

Electronic fuel injection systems used with petrol engines have been developed for about three decades. Single-point fuel injection did not last for long as it merely replaced the carburettor position and hence inherited all its shortcomings. Mixture preparation of the air and fuel injection at the inlet port has considerable effect on engine performance, fuel economy and emission characteristics. Therefore, enhancement of mixture formation before being inducted into the cylinder was a major concern of many investigators. In multi-point port injection, the most important parameters to be studied are the spray characteristic (spray angle, droplet size and injection timing), wall film formation and mixture formation above the inlet valves.

Kashiwaya et al [1] measured the spray angle by a photographic method. The droplet size is also measured at 10 cm below the nozzle tip which is assumed the average distance between the nozzle and the intake valves.

Their results on an actual engine test showed that fuel consumption rate decreases as the mean droplet size decreases. The dependence of the fuel consumption rate on the mean droplet size was comparatively high. The sprayed fuel of small droplet size contributes to a reduction in the fuel consumption and hydrocarbon levels and the enhancement of the startability at low temperatures. Fuel droplet impingement at the inlet port wall is found to be an important factor which contributed to the mixture preparation. With droplet injection systems, only a small fraction of the fuel spray is likely to evaporate during droplet flight, even with the engine hot.

The main advantage of using a narrow-angle fuel spray is that fuel could be made to impinge onto the back of the inlet valve, which is the hottest manifold surface. This could be the case with engines having one inlet valve. This enhances the evaporation of the liquid fuel. In engines with two inlet valves, the amount of the fuel impingement on the bifurcation between the valves is relatively high. Moreover, the velocity of the inducted air at the inlet ports is also low. This has the significant effect on the mixture formation in particular at low engine speeds and low loads.

Brown and Ladomatos [2] showed that only very small droplets with diameter less than 10 microns could follow the inlet air stream and eventual impingement onto the back of the inlet valves. These droplets represent usually less than 10 percent in volume of the total injected fuel although numerically it may well be over 80 percent in number counts. The size of the wall film formed by the impingement of the droplets has a strong influence on the mixture preparation. It affects the rate at which fuel evaporates from the film, the rate at which the film flows towards the inlet valve under the action of inducted air stream, and the size of the fuel droplets that are sheared off the film by the inlet air stream. Furthermore, air-fuel ratio excursions during the throttle opening are strongly influenced by the size of the fuel film.

Audi has adopted a multi-path induction manifold in their 100-2.8E V6 engine [3]. In this manifold, six-blade throttle valve, their opening and closing according to requirement is used. Thus each cylinder has two intake passages of different length and diameters; one for maximum torque and the other for maximum power. The six blade throttles are vacuum operated and are controlled by engine speed. Below 4000 rpm they remain closed to increase torque but above that speed they open completely for maximum power. The result is that more than 80 percent of peak torque output remains available at speed as low as 2000 rpm and as high as 5000 rpm. A multi-point injection and engine management system is incorporated with the engine.

Over the years there have been a number of investigations concerned with the mixture preparation process both in carburettor and fuel-injected SI engines. This included experimental investigations on fuel spray structure, the behaviour of fuel film, transportation and evaporation. Because the investigation is difficult, several investigators made us of theoretical models. Several models of the flow in an intake manifold have been proposed [4,5,6].

One simple manifold model that describes many of the airflow phenomena is the plenum or filling and emptying model. It is based on the assumption that at any given time the manifold pressure is uniform and it follows the continuity equation for airflow into and out of the intake manifold. For manifold pressures sufficiently low to choke the flow past the throttle plate, the flow rate is independent of manifold pressure. The airflow through the valve to each cylinder can be computed from the valve area, discharge coefficient and pressure difference across the valve, a sine wave function can be assumed. For calculating the manifold response to a change in load or throttling setting, simplifying assumptions can be made.

Lo and Lalas [7] studied the air, fuel vapour, and fuel droplet flow through an idealised manifold. The manifold used in their study was a straight circular pipe of 34.4mm in diameter and 30.5 cm long. Fuel was injected at the inlet of the manifold perpendicular to the main stream direction at zero initial velocity. One-dimensional, two-phase steady flow combined with evaporation and heat transfer through the manifold wall was assumed. The conservation equations and pertinent constitutive relations are solved numerically by the Bulirsh-Stoer integration scheme. A parametric study of exhaust gas recirculation, fuel type, initial droplet size, and air velocity was carried out. However, droplet deposition and hence liquid fuel film on the manifold wall was not included in this model.

Yun and Lo[8] restricted their model to the carburettor venturi. Droplet behavior moving in the carburetor-venturi was analysed by assuming one-dimensional, steady, two-phase flow. They used iso-octane as fuel in their model. Fuel evaporation and transportation were computed in terms of percent evaporation and droplet velocity respectively. Among the important parameters affecting droplet evaporation were the droplet size, droplet temperature, and the air temperature. Less significant parameters were air velocity, and air-fuel ratio. Their model however did not account for the fuel impaction on throttle plate and throttle-body.

A detailed analysis of both fuel and film evaporation is reported by Boam and Finlay [9]. This model was developed for an induction system with constant depression carburettor and manifold. They added a wall film uniformly distributed around the manifold. Although the mass flow rate of the fuel film was given, explicit values of its velocity and thickness were not reported. A multi-component gasoline based on its distillation curve was used as fuel in their model.

Milton and Behnia[10] reported a model similar to that of Boam and Finlay[9]. Sixteen equations of the steady state conservation of mass, momentum and energy were developed for liquid, vapor and air flow. Runge-Kutta method was used to solve these equations. The manifold was simulated as a rectangular duct of constant cross sectional area. A throttle plate in the form of a butterfly restriction was assumed. Fuel was injected upstream of the throttle plate at a distance of 122mm. Droplet deposition on the manifold wall was identified based on Von-Karman's extension of Reynolds Analogy. According to this analogy, eddy diffusivity moves a

proportion of droplets from the turbulent core and deposits them close to the wall within an effective stopping distance. While this approach was developed for solid particles, Boam and Finlay also used it for fuel droplet flow. In this model, film thickness, its velocity and percentage of evaporation at different throttle openings were computed.

A different approach was adopted by Servati and Yuen[11]. They used the equations of motion developed by Sjenitzer [12] for a droplet in a two-dimensional flow under the action of aerodynamic drag. They improved on the model by including the effect of the gravitational force on the droplet flow. The equation for droplet evaporation in a moving gas stream developed by Ranz and Marshall [13] was used in this model. An idealized manifold of circular cross section was used. A number of droplets of equal size were introduced at the beginning of the manifold with zero initial velocity. Droplet evaporation and acceleration were examined. Droplet deposition was identified with respect to the distance that the droplet has moved in the transverse direction to the flow. These allowed the percentage of total fuel deposition on the wall to be determined. Assuming couette flow for the film on the bottom half of the pipe, film velocity was determined from the shear force at the gas-film interface. This study was carried out for the initial droplet sizes ranging from 100 and 300 microns and for gas velocities ranging from 1 to 5 m/s. However, throttling was not considered in this model. Furthermore, the reported size of the diameter of manifold, 15.3 cm is far from the normal diameter of conventional manifolds. They however, reported the percentage of droplet deposition for different manifold geometries (length/diameter).

Brown and Ladommatos[14] presented a computer simulation of mixture preparation in the inlet port of a port-injected spark-ignition engine operating at low load and low speed. The equations of fuel droplet evaporation and motion were used to study droplet deposition, wall film evaporation, and wall film flow along the manifold walls. Using this model they investigated the effects of manifold temperature and air temperature on spray droplet size, injection timing and fuel volatility. Their model is based on steady air flow and single-component fuel.

COMPUTER MODELLING OF MODIFICATIONS

A study of the air-fuel flow interactions in the port injection fuelling system was carried out for a number of new modifications which have been developed A two-dimensional two-phase model of the flow in the induction system of a port injected SI engine employing the Fluent software was first carried out to benchmark modified behavior (Hakim et al [15]). In that work, the air flow fields coupled with fuel spray injection are simulated at engine speeds of 800, 1500 and 3000 rpm. In all those cases the initial droplets at the injector nozzle are taken as 80 microns and initial jet velocity is 27 m/s with an angle of 22.7 degrees. The maximum air steam velocity was found to occur at the valve seat at 62 m/s at 800 rpm, 76 m/s at 1500 rpm and 98 m/s at 3000rpm. Fuel spray propagation is found to be affected by air flow

fields as fuel droplet experienced deceleration and diverted towards the upper side of the inlet port. As engine speed increased, air velocity in the inlet port increases and hence the fuel spray tends to become narrow as it impacts on the inlet port wall.

In the context of designing the modifications, the aim was to enhance the air-fuel mixing and to divert the fuel spray stream from the port wall. Four techniques have been modeled using Fluent V4.2 software and with boundary conditions for the air stream flow and fuel injection at engine speeds 800, 1500 and 3000 rev/min. The parameters which are studied after time 2 millisecond after injection of the fuel droplets are the radial velocity component and position of the fuel. It is observed that most fuel droplets from the spray ligament near the upper side of the inlet port have a high tendency to impinge on that wall. Therefore any technique which increases the air stream velocity near that wall will have an advantage in diverting these droplets toward the core of the flow.

Orifice-Type Modification (modification 1)

This is an orifice-like shape in a form of a nozzle which could be placed on the upper half of the inlet port. By doing so, a high speed sheet of air flows near the wall could be achieved even at low engine speeds. The dimensions and the location of the curved plate is show in Fig (1a) at a position just above the area where the spray impinged.

Flapper Device Technique (modification 2)

This is an adjustable flapper which is placed at the upper half of the cylinder as shown in Fig. (1b). The device is linked with the accelerator pedal that controls the opening of the inlet manifold (engine speed). The technique uses an adjustable device which corresponds to different engine speeds since the main problem inherited is a low engine speed. As the engine speed decreases, the flapper will be activated. However in this study, the position of the flapper is kept constant for engine speeds of 800, 1500 and 3000 rpm. The flapper is positioned at a distance of 21.7 mm from the injection point. The maximum distance from the flapper tip to the inlet port wall is 3.5 mm.

Sliding Plate with Separator Technique (modification 3)

An adjustable sliding plate with a separator is modeled. The dimensions and location of this technique is shown in Fig (1c). It is located at the inlet manifold before the injection point. The sliding plate can be adjusted corresponding to the engine speeds. In this study, the distance of the opening from the separator is 4.4 mm, 7.3 mm and 10.2 mm when simulated for engine speeds of 800,1500 and 3000 rpm, respectively. The purpose of the sliding plate and the separator is to block and separate the flow field inside the intake manifold and the intake port so that the upper side of the manifold will have the high air flow. This high air flow has a high tendency to avoid fuel impingement at inlet port wall.

Sliding Plate Type Modification (modification 4)

In this technique, the sliding plate is used without the separator. The dimensions and location of the sliding plate is shown in Fig.(1d). The sliding plate is an adjustable device which is being linked with the accelerator pedal that controls the engine speed or with an actuator connected to the inlet manifold. The height of the sliding plate is 14.5 mm, 7.3 mm and 4.4 mm at engine speeds of 800, 1500 and 3000 rpm respectively

NUMERICAL RESULTS OF THE COUPLED PHASE

Results from the two-phase two-dimensional CFD simulations show that the use of orifice (modification 1) has some effect on shifting their flow towards the lower part of the port compared with the spray with a normal manifold (Fig.2a). Similar behavior is seen in modification 2 which employs the flapper for all engine speeds (Fig.2b). For the sliding plate with separator (modification 3), about 10% of the spray are diverted towards the cylinder at engine speed of 800 and 1500 rpm. As the engine speed increases to 3000 rpm, all the fuel spray in the intake port impinged onto the port wall with a narrow shape. These could be attributed to the dynamic effect of the very high speed air at the lower fringes of the spray (Fig.3a). Fig.(3b) presents the spray flow coupled with the air flow for the sliding plate technique (modification 4). The fuel is only diverted towards the cylinder at an engine speed of 1500 rpm. However, at 800 rpm, fuel is forced to impinge onto the port wall by the high speed air flow since the rate of vaporization of the droplets is low.

A comparison of the performances of the four techniques may be achieved via examining the radial and vertical droplet velocity components, two milliseconds into the flow. Fig.(4a) and Fig.(4b) show further the characteristic of droplet behaviour in the x- (radial) and y-(towards the cylinder-head) directions. It can be seen that the droplet velocities in the x-direction for the upper fragments of the spray move faster than the lower fragments. This proves that the possibilities of droplets to impinge on the upper part of the port wall are very high. Modifications 1 and 2, however shows that the velocity in x-direction is significantly reduced compared with the normal induction system and the other modifications in particular of those spray segment near the upper inlet port wall. This is because of the drag force effect on the droplets is very high thus decelerating the velocities of the droplets in this direction.

COMPUTER MODELLING FOR DIRECTION INJECTION

An appropriate development to the above work is naturally the considerations for gasoline direct injection and the behavior of fuel sprays coupled with air within the confines of the combustion chamber. Prior to this study, an experiment on the behavior of fuel sprays under ambient

conditions has been reported by Saqaff et al [16]. The behavior of fuel spray from an intermittent fuel spray produced by the injection system of a WIRA 1.6L engine was studied. Employing a Laser Imaging System and the Phase Doppler Anemometer system (PDA), fuel droplet size and velocities in the radial and axial direction were determined when sprayed to ambient atmospheric conditions. Results of the study revealed that the spray angle at all engine speeds is about 36.8 degrees, the core of the spray characterised by dense fuel droplets which penetrated a distance of about 18 cm. At all engine speeds, droplet diameters ranges from 65 to 80 microns, with a tendency to increase towards the outer boundary of the spray. Droplet velocity in the direction of spray is found to range between 1.4 to 5.5 m/s depending on the size of the droplets whilst in the radial direction, it is 0.1 to 1.4 m/s.

Based on the above findings, a three-dimensional two-phase model was developed to determine the behavior of fuel droplets injected directly into the cylinder aft of the inlet valve. The modeling technique and results of the study have been reported in a paper by Ow et al [17]. A body fitted structured grid of the inlet port, inlet valve (partly open), cylinder and piston head was generated using 14,280 cells. The piston is assumed to be 17.5 mm from the cylinder head in a suction stroke period. Air flow velocities in the three directions at engine speeds of 1,500 and 3000 rpm are computed with the required boundary conditions. Fuel droplets of sizes 50 microns at the nozzle tip are simulated and injected over a spray angle of 22.7 degrees relative to the axial direction of the spray. Results from the study revealed that the fuel injector angle has to be offset relative to the valve stem axis. It was found that off-setting the injection by about 30 degrees with the inlet valve stem achieves significant spray distribution in the cylinder. In all cases, it was found that a large portion of the spray impinged on the piston head, without modifications to the piston head to generate swirl. Fuel droplet velocity is observed to be less than 2 m/s. It is also found that the fuel droplets velocities at the core of the spray are lower than those at the outer perimeters.

Based on the above studies, modifications were made initially to the piston head and subsequently to the cylinder head as well for a three-dimensional two-phase flow study. This is aptly reported by Ow & Saqaff[18] some of which results are reproduced here for completeness.

MODELING METHODOLOGY

Direct fuel injection in petrol engine is investigated through a three-dimensional, two-phase computer model. FLUENT software is used to model the air flow and the fuel spray propagation. The cylinder and induction system geometry is that based on a WIRA 1.6 L engine. A body-fitted structured grid of the inlet port, inlet valve, cylinder and piston head with 14.280 cells has been used. The piston is modeled to be 17.5 mm from the cylinder head in a suction stroke period. Air flow velocities (x-, y- and z-components) corresponding to engine speeds of 1,500 rpm and 3000 rpm

are computed with the required boundary conditions. One feature of FLUENT is that one could select the mean droplet size to be injected into the cylinder. Hence an injection of 50 micron droplets at the nozzle tip is selected with a spray angle of 22.7 degrees and the spray nozzle angles at 90, 120 and 150 degrees with respect to the x-z plane. Different spray locations, nozzle angles and engine speeds are studied. Fig.(5) illustrates the geometry modeled for flow in the intake system and combustion chamber. The unmodified piston is assumed to be 17.5 mm from the cylinder head in a suction stroke period.

Three different configurations concerning the shape and design of the piston and combustion chamber are investigated. The direct injection behaviour inside the cylinder with a normal piston (flat type piston head), a modified piston head (with a cavity on one side) and a modified cylinder head are studied employing the above positions and angles of spray nozzle. The fuel and the air velocity distribution in the cylinder are studied through three different positions of the piston towards the end of the compression stroke as well. Hence the distance from the top dead centre (TDC) to the piston head is set at 60, 40 and 20 mm. Boundary conditions related to engine speed at 2000 rpm are employed in these simulations.

RESULTS AND DISCUSSIONS

Typical results of suction stroke injection at engine speed of 3000 rpm are presented in the following figures. Airflow velocities and fuel spray development have been computed through the inlet port, inlet valve seat and inside the cylinder.

Fig.(6) illustrates the air velocity distribution at engine speed of 3000 rpm. Maximum air velocity is observed to be 152.4 m/s at the exit of the inlet valve. It is possible to observe that there is high velocity air dominant towards the right hand side of the valve, which occupied almost half of the cylinder. This high velocity air stream with high turbulent flow could be useful in mixing the fuel spray with air inside the cylinder during the suction stroke. Based on this observation and the nozzle location study, it is decided that the fuel nozzle will best be located on this side of the valve. However the exact angle of injection will be decided on the results of the spray model.

Fuel spray was modeled with injection angles of 90 degrees 120 degrees and 150 degrees with respect to the x-y plane (Figs 7 to 11) and at engine speeds of 1500 rpm and 3000 rpm. It is to be noted that the angle of the spray injection is studied with respect to the x-z plane. It is found that at low engine speeds the spray penetrated towards the piston head with little deflection. However at higher engine speed (3000 rpm) the spray deflection towards the cylinder wall is clearly evident where the injection angle is set at 90 degrees with the x-z plane (Figs. 7&8) This due to high air velocities in a swirl motion, towards the cylinder side and piston head. In actual running engine, the piston is moving down towards the bottom dead centre when

injection takes place. This may reduce the impingement of fuel spray on the piston head

In Figs. (9&10), the fuel injection angle is 120 degrees directed towards the valve shoulder and the results show that at high engine speed of 3000 rpm, fuel spray seems to be totally affected by the air stream past the inlet valve. Fig. 11 shows the spray development then the fuel is injected at an angle of 150 degrees at engine speed of 3000 rpm. The spray shows a tendency to spread over a large area with little impingement on the piston head. It is found that at engine speed of 1500 rpm about 65% of the fuel impinges on the valve shoulder.

A further study was carried out to investigate the behaviour of the air-fuel inside the cylinder after the induction stroke. In this case the direct injection is taking place in the compression stroke (as the case in CI engines) at an engine speed of 2000 rpm. The piston is located at three different positions from the top dead centre 20, 40 and 60 mm respectively. Three different configurations were investigated. Those are the normal flat piston head, modified piston head and the modified combustion chamber. Figs.(12&13) illustrates the velocity vector of the air and the particle droplet trajectories of the fuel (the piston is at 20 mm from the top dead centre at crank angle 51 degrees). It is clear that the velocity is high at the piston head and the air is directed towards the right side making the fuel stay near the inlet valve. The same behaviour was noticed for the other cases of piston positions at 40 and 60 mm from the top dead centre. However these results indicate the inefficient mixing of fuel and air inside the cylinder. To enhance the mixing process, the piston was modified. Fig. 14 shows the velocity magnitude and the droplet trajectories. It is clear that fuel spread is due to the swirl effect controlled by the shape of the piston leading to more effective mixing. A single vortex is formed in the centre, it is found that this vortex spread as the piston moves towards the end of the compression stroke.

The modified cylinder head show better improvement in the air-fuel distribution in the cylinder, but is likely that the modification of the piston has more significant effect. Fig. 15 illustrate the velocity vector of the air and the fuel, it can be noticed that a higher velocity is taking place at the piston head with an improvement of fuel distribution compared with the case of an unmodified cylinder head. Figs.(16&17) show the velocity at two different slices away from the centre of the cylinder. It is clear that the velocity is higher near the centre providing optimum mixing of the charge around the spark plug

CONCLUSIONS

Electronic fuel injection systems used with petrol engines have been developed for about three decades. Single-point fuel injection did not last for long as it replaced the carburetor positon and hence inherited its shortcomings. The main objective has been to understand the behaviour and the characteristic of the air-fuel flow through the induction system and

in-cylinder. The two-dimensional two-phase computer model which studied the effect of various modifications on propagation of spray at different engine speeds provided the following conclusions. The maximum air velocity has increased by about 7% when using the modification compared with unmodified inlet port. The increase in air velocity has significant effect on the air-fuel mixing in particular at low engine speeds. It is found that an air-sheet (20 m/s) is generated by the orifice device near the upper side of the inlet port. Modifications 1 and 2 have significant effect in shifting the fuel droplet towards the lower part of the inlet port. This is clear as the fuel droplets velocity in the vertical direction of the fuel segments near the upper part of the inlet ports show a tendency to be low at all engine speeds.

In direct fuel injection, fuel is injected into the cylinder during the suction stroke. Therefore, problems related to fuel deposition and cyclic combustion variation will be eliminated completely. Direct fuel injection in a three-dimensional, two-phase computer model study employing FLUENT software suggested the correct position of the nozzle is towards the right hand side of the inlet valve. Fuel spray is modeled at two locations, 15mm and 20 mm (away from the inlet valve) with injected angles 90,120,150 degrees respectively with respect to the x-z plane. Base on the result so the study, maximum air velocity is found to be at eh exit of the inlet valve. There are two highly turbulent domains observed on both sides of the inlet valve and a stagnant domain directly below the valve face. The fuel injection angle has to be offsetted relative to the valve stem axis. It is found that offsetting the injection by 30 degrees relative to the inlet valve stem gives more significant spray distribution in the cylinder.

Direct injection inside the cylinder during the compression stroke reveals that for flat piston head, about 70 percent of the fuel spray move towards the right side of the inlet valve leading to poor mixing of the fuel and air. As the piston moves up towards the top dead centre, the air velocity increases as noticed in all the cases studied. Results for the modified cylinder head with a normal piston show a slight better air-fuel distribution. It was noticed that piston shape has more significant effect on the velocity distribution compared with the other cases.

ACKNOWLEDGEMENT

The research for this paper was conducted under sponsorship of the IRPA fund, project number 50032. The authors gratefully acknowledge the support of the BRC, UiTM Shah Alam and the Faculty of Mechanical Engineering.

REFERENCES

Kashiwaya, M., Kosuge, T., Nakagawa, K and Okamoto, Y.,*The effect of atomization of fuel injectors on engine performance,* SAE paper 900261, 1990.

Brown, C. N., and Ladommatos, N.,*The effects of mixture preparation and rapid residual on the performance of a spark-ignition engine with air-shrouded port injectors at low load and low speed,* Proc. Inst. O Mech. Engrs., IMechE, London, Vol 206, pp17-30, 1991.

Auto-international, Vol21, No. 5, May 1994

Trayser, D A, Creswick, F A, Giesike J A, Hazard, H R, , Weller, A E and Locklin, D W, *A study of the influence of fuel atomization, vaporization and mixing processes on pollutant emissions from motor- vehicle powerplants.*Battelle Memorial Inst. Columbus, Ohio, 1969

Tabaczysnki, R J, *Effects of inlet and exhaust system design on engine performance,* SAE paper 821577, 1982

John B Heywood, *Internal Combustion Engines Fundamentals* Mc Graw Hill Publishing Company

Lo, R S and Lalas, D P, *Parmetric study of fuel droplet flow in an idealized automotive engine induction system,*SAE Trans., 770645, pp 2398-2413, 1977

Yun, H J and Lo R S, *Theoretical studies of fuel droplet evaporation and transportation in a carburettor venturi,* SAE paper No. 760289, 1976

Boam, D J and Finlay, I C, *A computer model for fuel evaporation in the intake system of a carburetted petrol engine,* Proc. Instn. Mech. Engrs Conf. C89/79, pp25-37, 1979

Milton, B E and Behnia, M, *Numerical study of the interchanging vapour, droplet and film flows in gasoline engine manifold,* Proc. Instn. Cent. Heat Transfer, Ser. 26,pp245-258, 1989

Servati H B and Yuen, W W, *Depositon of fuel droplets in horizontal intake manifolds and the behaviour of fuel film flow on its wall,* SAE Trans., 840239, pp 125-133, 1984.

Sjenitzer F, *The evaporation of a liquid spray injected onto a stream of gas,* Chemical Engineering Science, Vol 17 p 309-322, 1962

Ranz W E and Marshall W R Jr, *Evaporation from drops,* Vol 48, 1952.

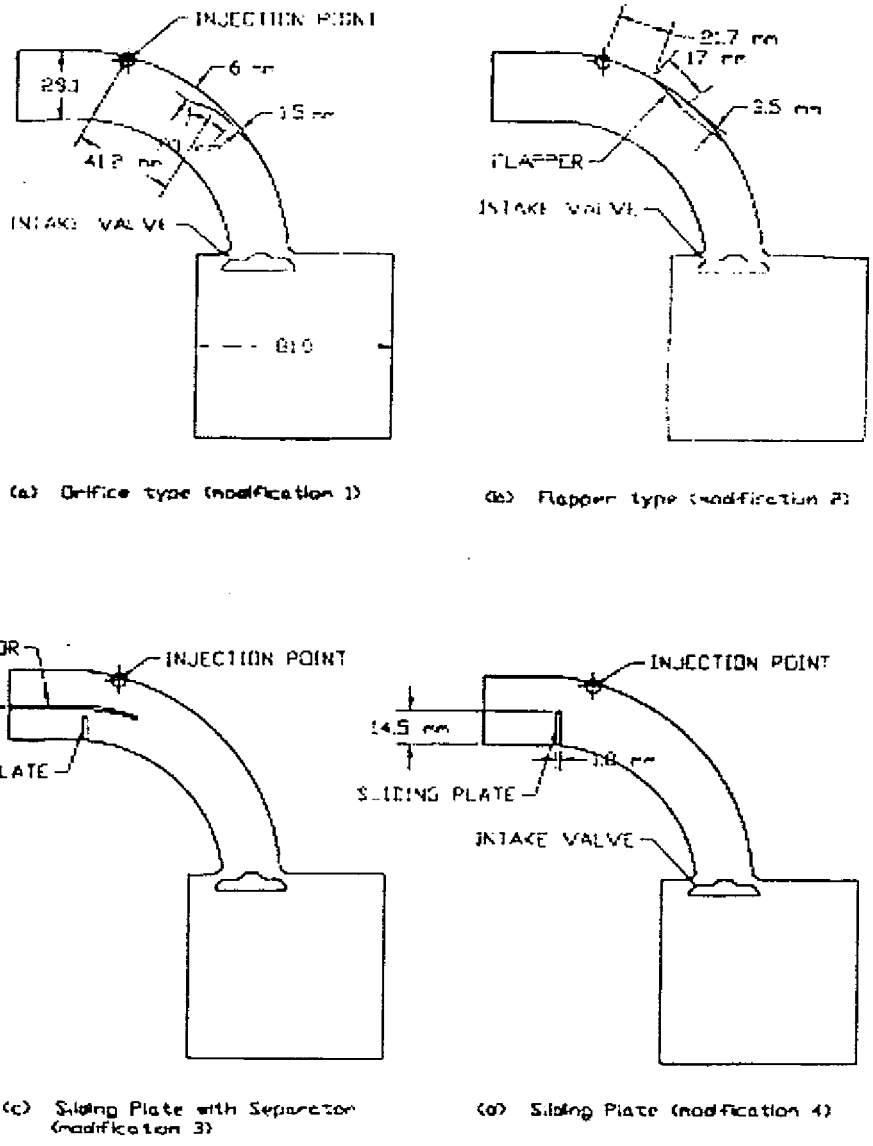
Brown, C N and Ladommatos, N, *A Numerical study of fuel evaporation and transportation in the intake manifold of a port injected spark-ignition engine,* Proc. Instn Mech. Engrs, Vol. 205, pp161-175, 1991

Hakim, A Abbas, OW C S, Abd Ghani Ujang, Ahmad Fakri Shaari, *Two-dimensional two-phase flow analysis in port injected spark ignition engines,* Proc. MSTC 97, Vol II, pp 43-50, 1997

Saqaff A Alkaff, Hakim A Abbas, OW C S, *Analysis of Fuel Spray Characteristics of an Electronic Fuel Injection,* Proc. MSTC 98, Symposium A, pp 72-79, 1998

Ow C S , Hakim A Abbas, Saqaff, A Alkaff, *Direct (In-Cylinder) Fuel Injection Strategy for Spark Ignition Engines: Three-Dimensional Two-Phase Computer Simulations,* Proc. MSTC 98, Symposium A, pp 80-86, 1998.

OW CS & Saqaff A Alkaff, *Direct Fuel Injection Strategy for Spark Ignition Engines,* Proc. Asia-Pacific Conf. On the Built Environmnet, Taiwan, pp B8-1 to B8-15, 1999.



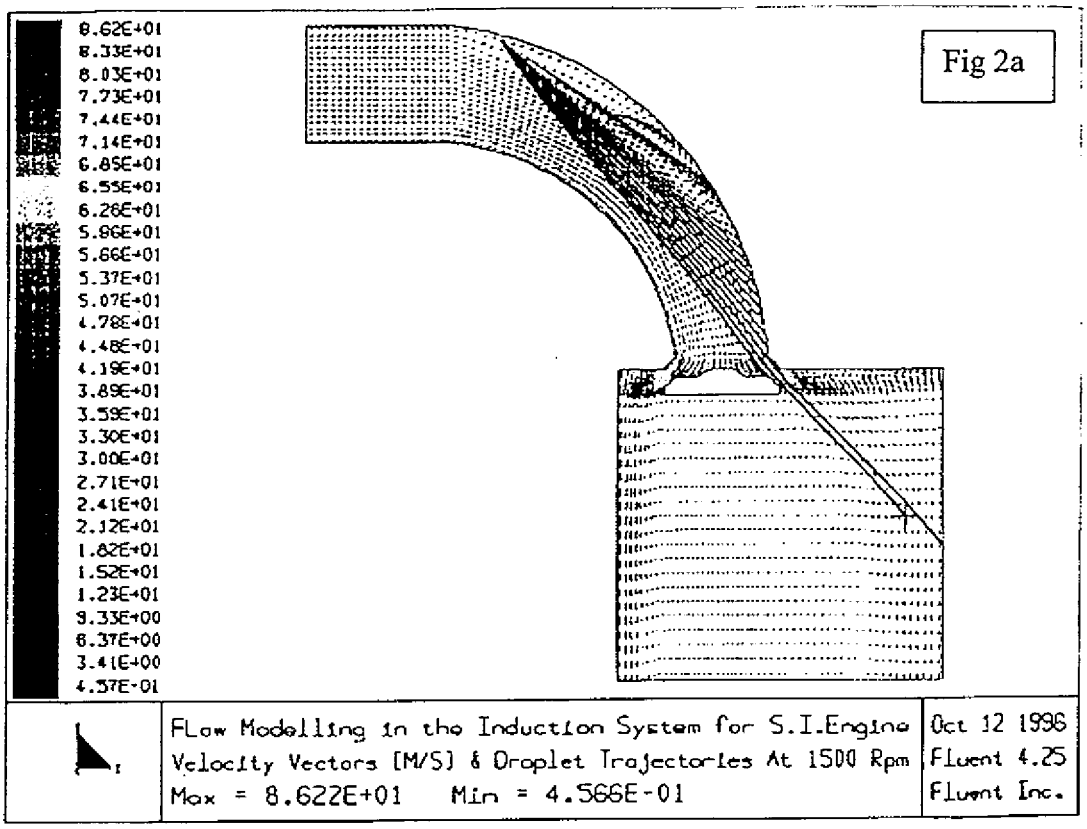
(a) Orifice type (modification 1)

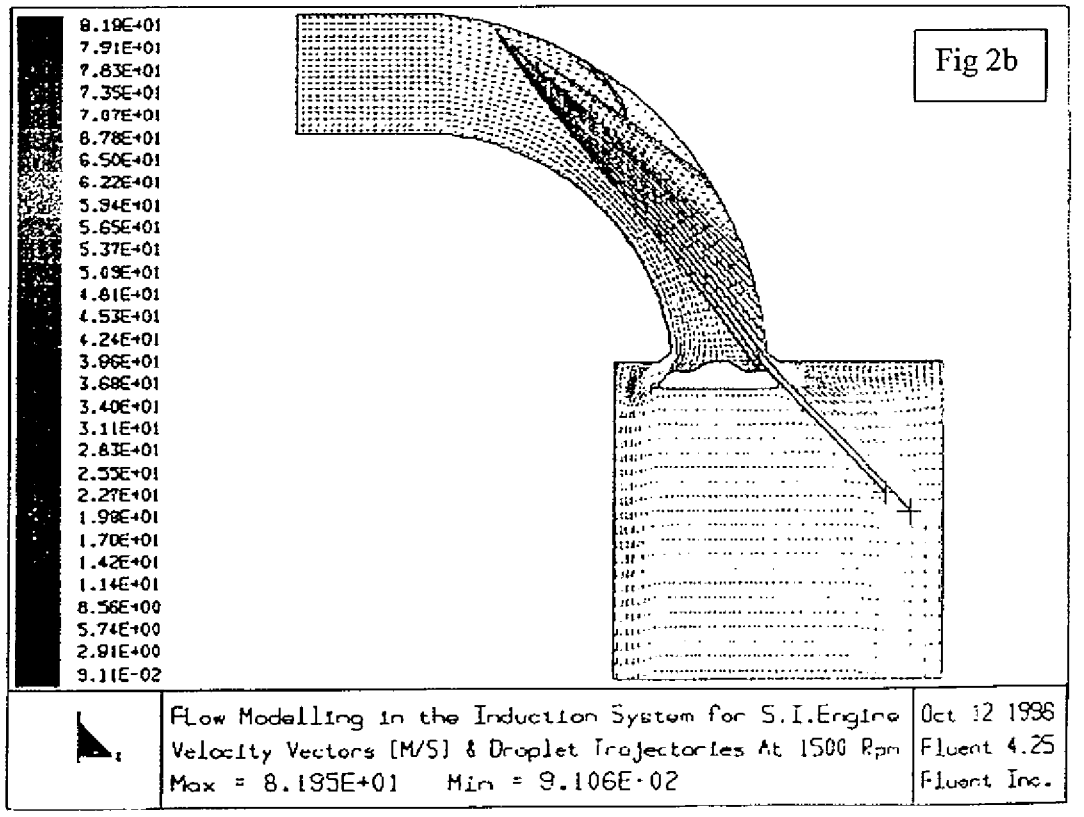
(b) Flapper type (modification 2)

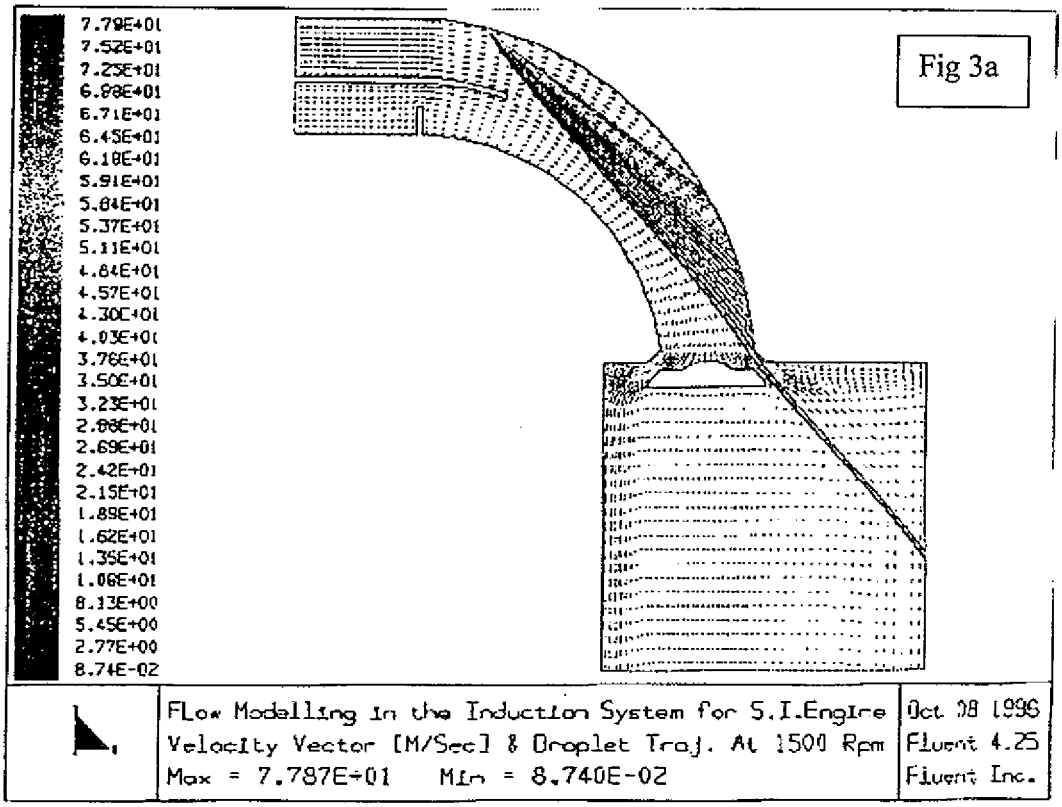
(c) Sliding Plate with Separator (modification 3)

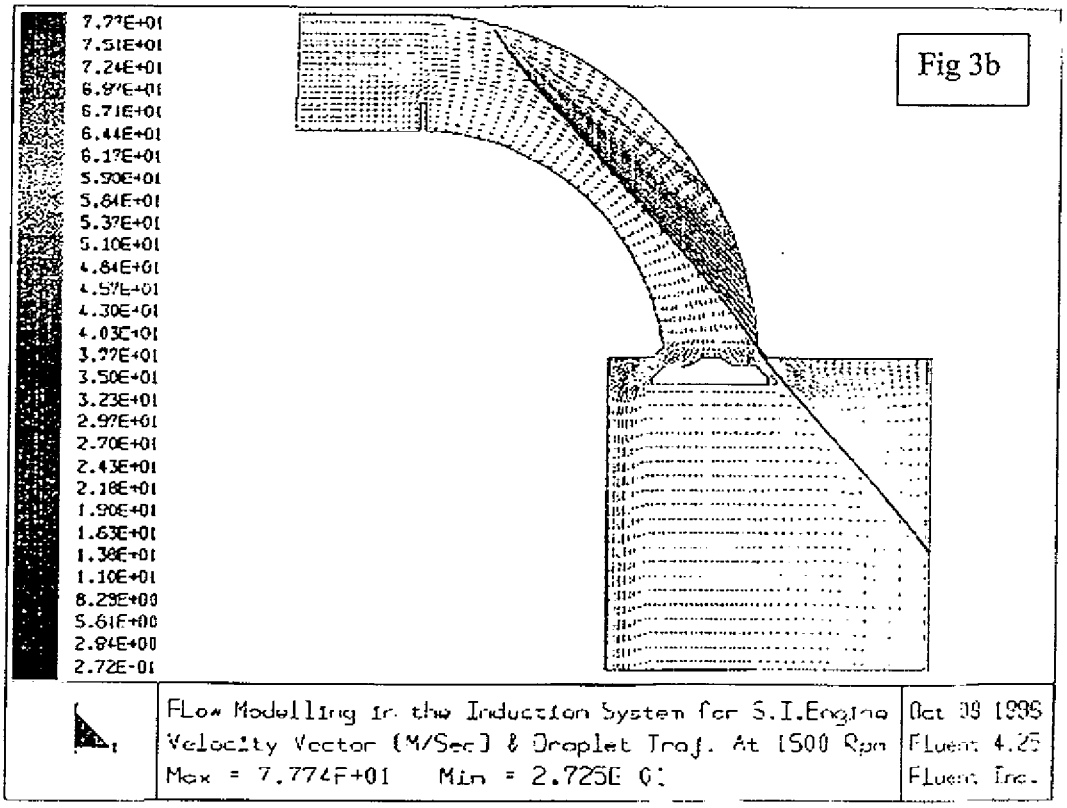
(d) Sliding Plate (modification 4)

Figure 1 Geometry of the new modifications to the induction system in spark ignition engines


8.62E+01
8.33E+01
8.03E+01
7.73E+01
7.44E+01
7.14E+01
6.85E+01
6.55E+01
6.26E+01
5.96E+01
5.66E+01
5.37E+01
5.07E+01
4.78E+01
4.48E+01
4.19E+01
3.89E+01
3.59E+01
3.30E+01
3.00E+01
2.71E+01
2.41E+01
2.12E+01
1.82E+01
1.52E+01
1.23E+01
9.33E+00
6.37E+00
3.41E+00
4.57E-01
Fig 2a
Flow Modelling in the Induction System for S.I.Engine
Velocity Vectors [M/S] & Droplet Trajectories At 1500 Rpm
Max = 8.622E+01 Min = 4.566E-01
Oct 12 1996
Fluent 4.25
Fluent Inc.


8.19E+01
7.91E+01
7.63E+01
7.35E+01
7.07E+01
6.78E+01
6.50E+01
6.22E+01
5.94E+01
5.65E+01
5.37E+01
5.09E+01
4.81E+01
4.53E+01
4.24E+01
3.96E+01
3.68E+01
3.40E+01
3.11E+01
2.83E+01
2.55E+01
2.27E+01
1.98E+01
1.70E+01
1.42E+01
1.14E+01
8.56E+00
5.74E+00
2.91E+00
9.11E-02
Fig 2b
Flow Modelling in the Induction System for S.I.Engine
Velocity Vectors [M/S] & Droplet Trajectories At 1500 Rpm
Max = 8.195E+01 Min = 9.106E-02
Oct 12 1996
Fluent 4.25
Fluent Inc.


7.79E+01
7.52E+01
7.25E+01
6.98E+01
6.71E+01
6.45E+01
6.18E+01
5.91E+01
5.64E+01
5.37E+01
5.11E+01
4.84E+01
4.57E+01
4.30E+01
4.03E+01
3.76E+01
3.50E+01
3.23E+01
2.96E+01
2.69E+01
2.42E+01
2.15E+01
1.89E+01
1.62E+01
1.35E+01
1.08E+01
8.13E+00
5.45E+00
2.77E+00
8.74E-02
Fig 3a
FLow Modelling in the Induction System for S.I.Engine
Velocity Vector [M/Sec] & Droplet Traj. At 1500 Rpm
Max = 7.787E+01 Min = 8.740E-02
Oct 08 1996
Fluent 4.25
Fluent Inc.


7.77E+01
7.51E+01
7.24E+01
6.97E+01
6.71E+01
6.44E+01
6.17E+01
5.90E+01
5.64E+01
5.37E+01
5.10E+01
4.84E+01
4.57E+01
4.30E+01
4.03E+01
3.77E+01
3.50E+01
3.23E+01
2.97E+01
2.70E+01
2.43E+01
2.16E+01
1.90E+01
1.63E+01
1.36E+01
1.10E+01
8.29E+00
5.61E+00
2.94E+00
2.72E-01
Fig 3b
FLow Modelling in the Induction System for S.I.Engine
Velocity Vector [M/Sec] & Droplet Traj. At 1500 Rpm
Max = 7.774E+01 Min = 2.725E 0
Oct 08 1996
Fluent 4.25
Fluent Inc.

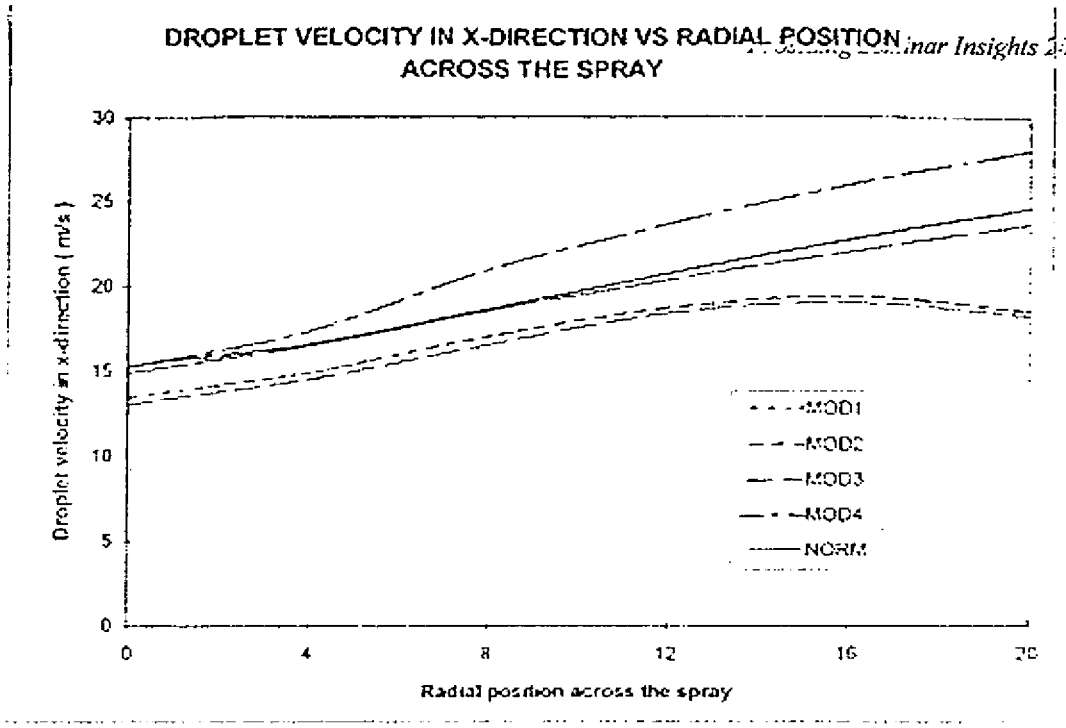


Figure (4a) Comparison between the droplet velocity in y-direction for all modifications at engine speed of 800 rev/min

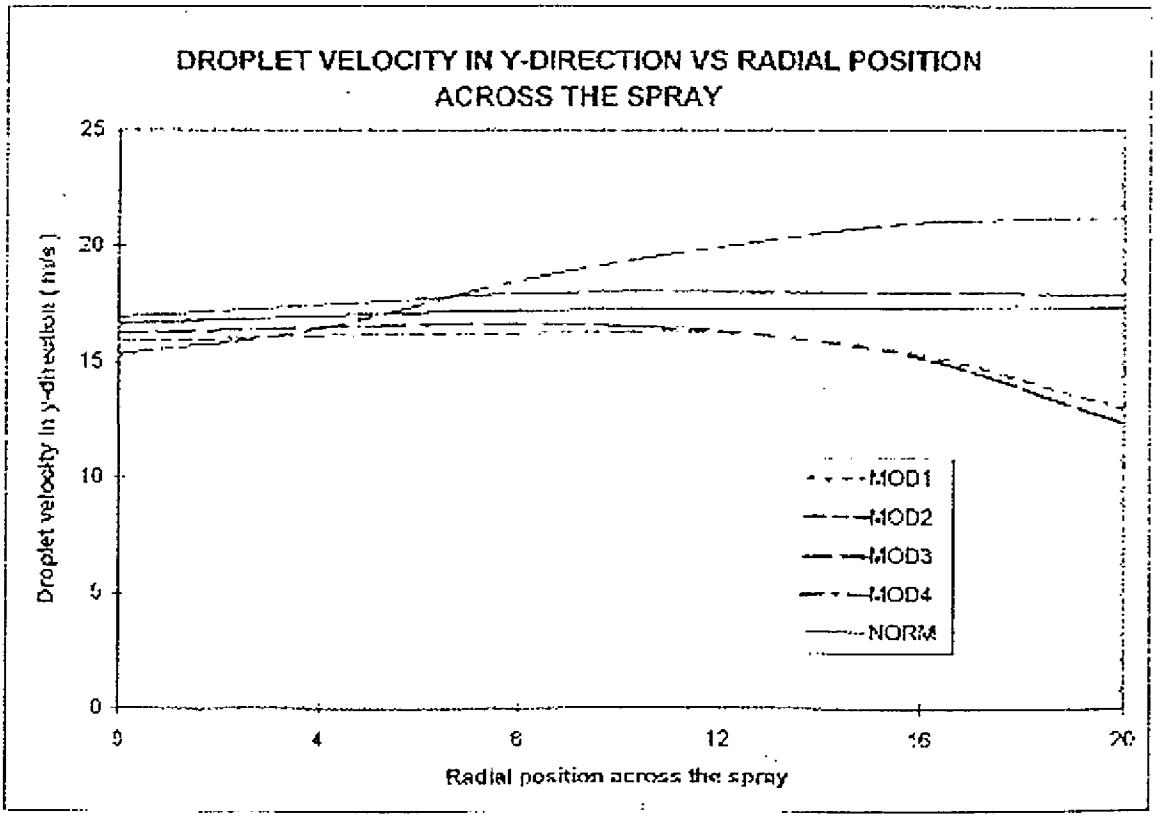


Figure (4b) Comparison between the droplet velocity in y-direction for all modifications at engine speed of 800 rev/min

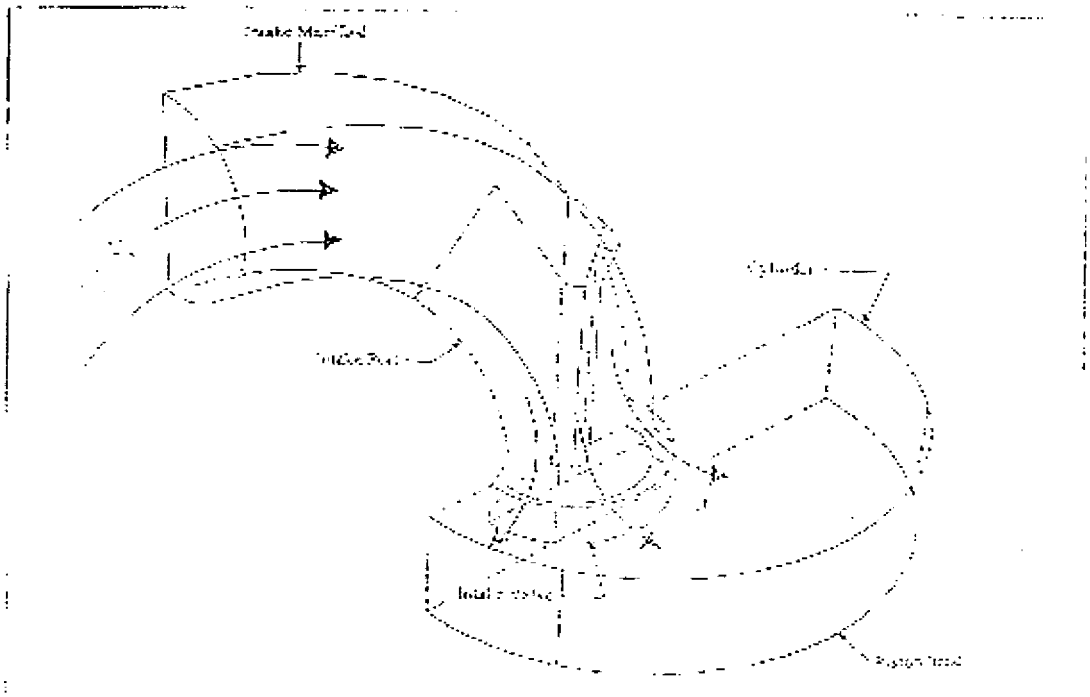
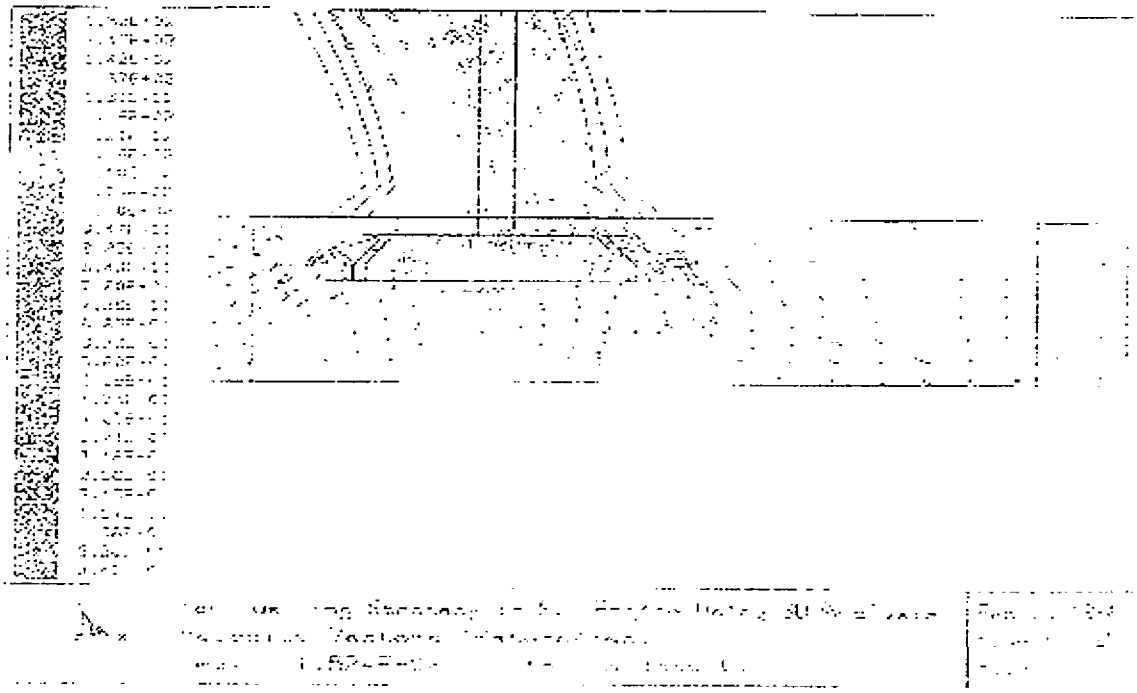


Figure (5)Three-dimensional airflow model.



Zooming view of airflow velocity vector (engine speed of 3000 rpm).

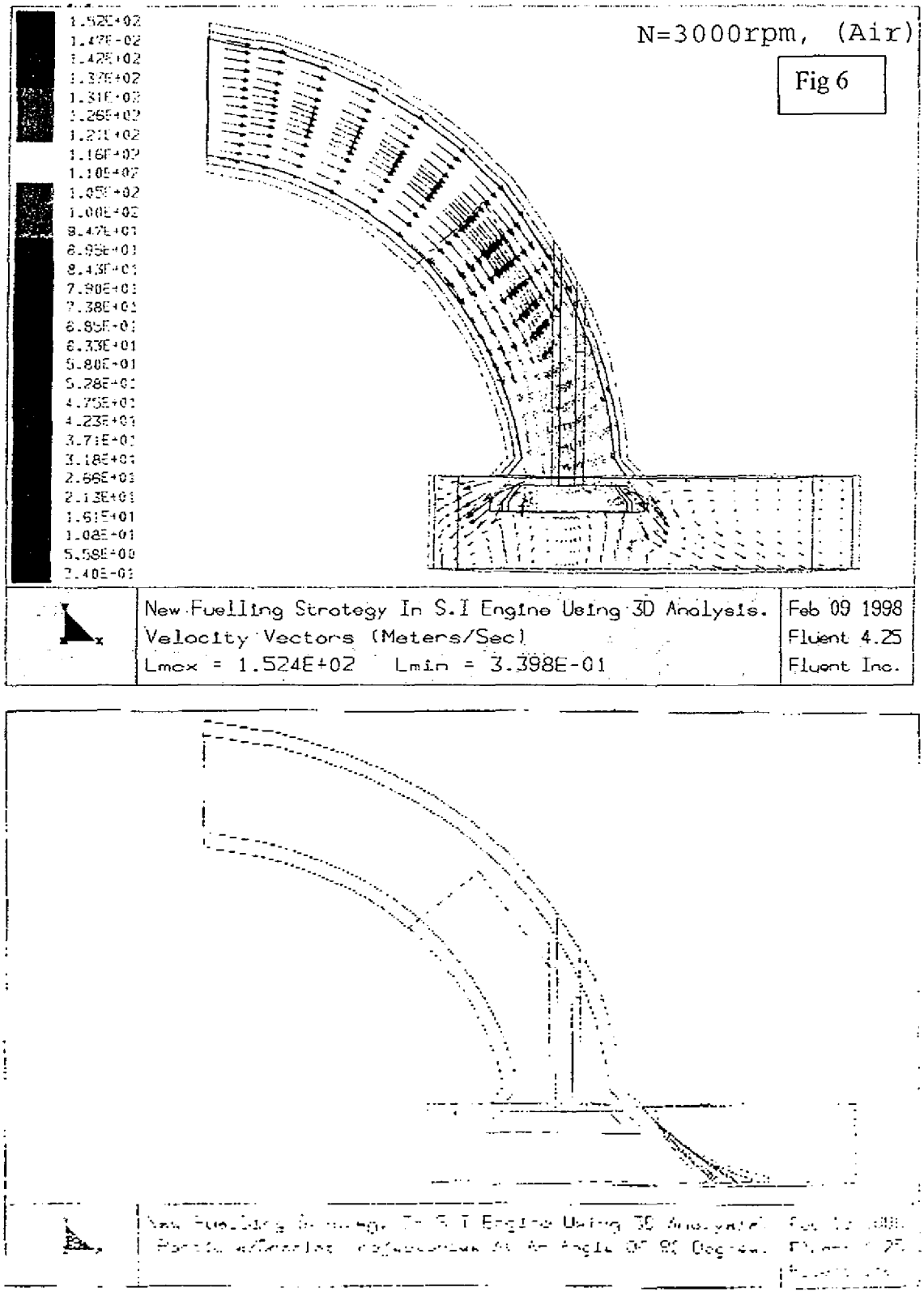


Figure (7) Front view of the droplet trajectories (engine speed of 3000 rpm).

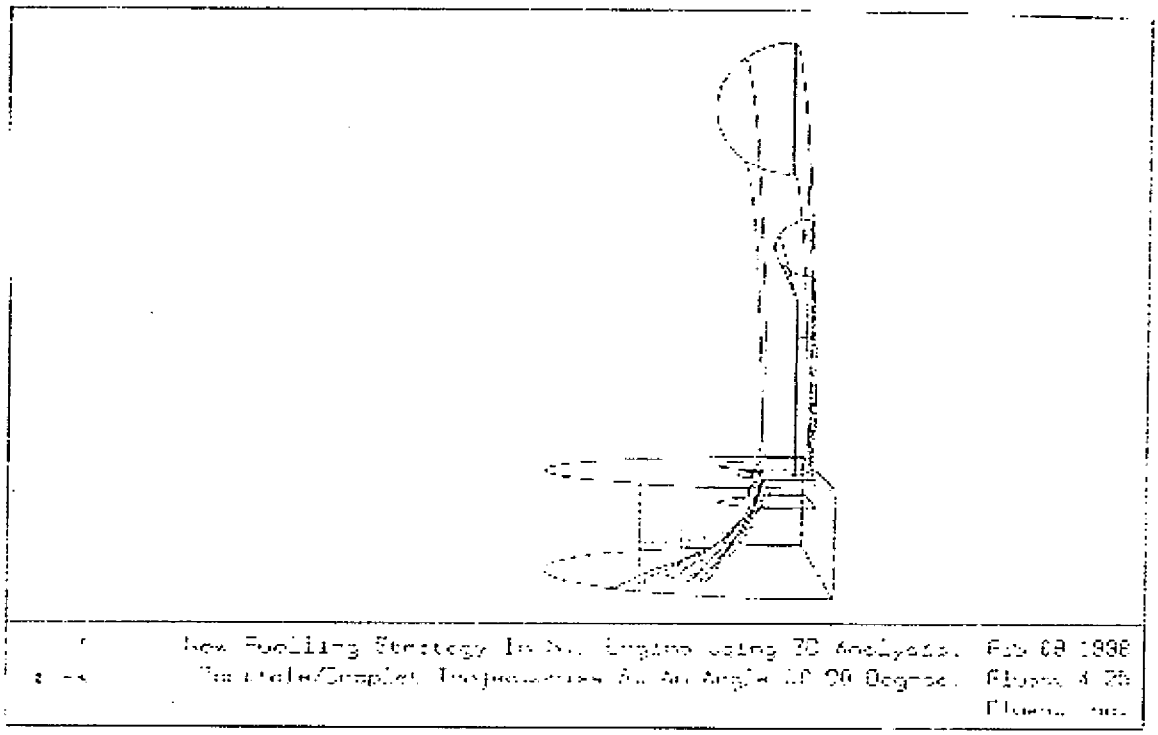
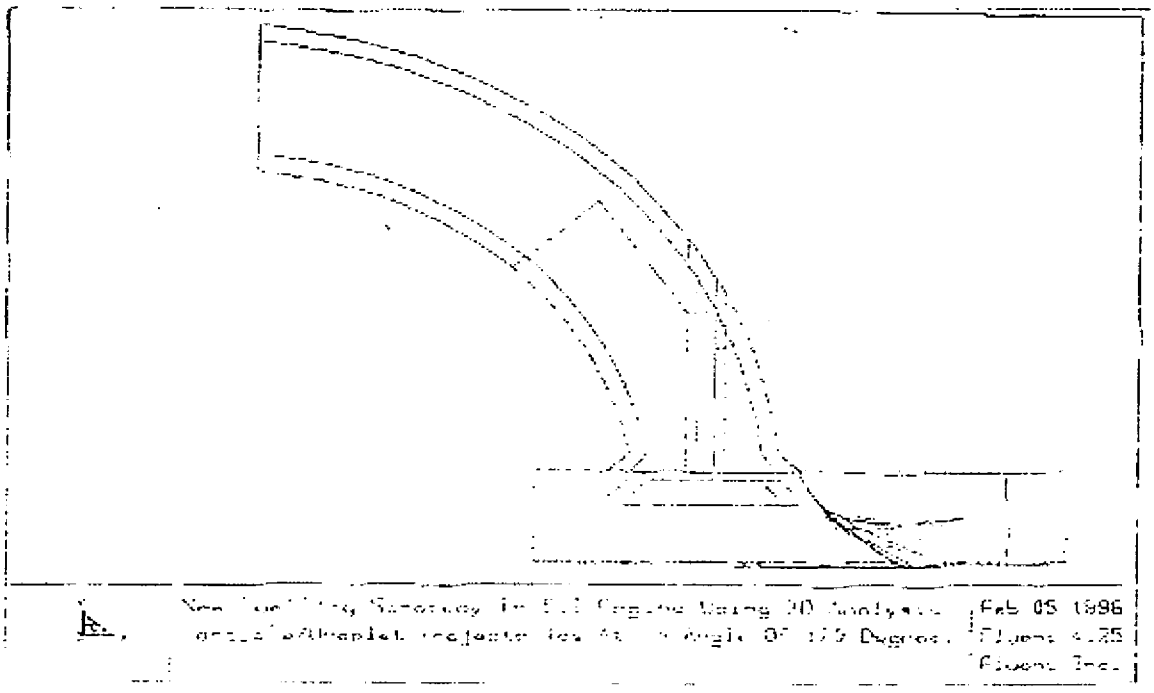
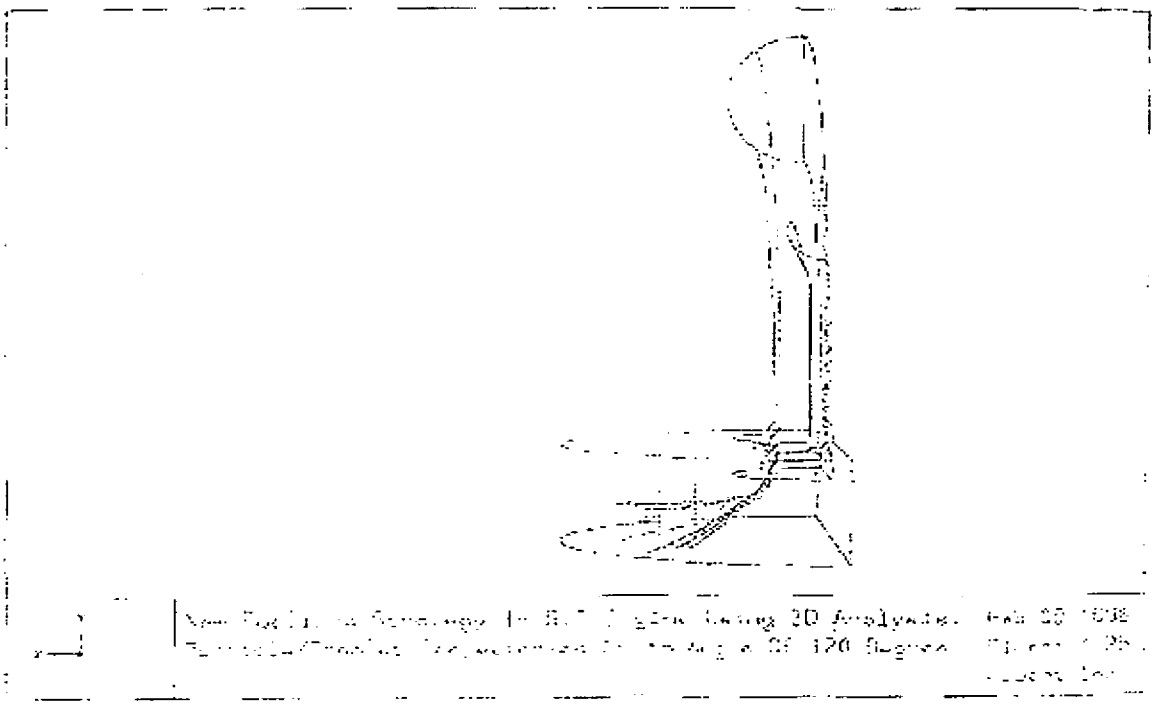


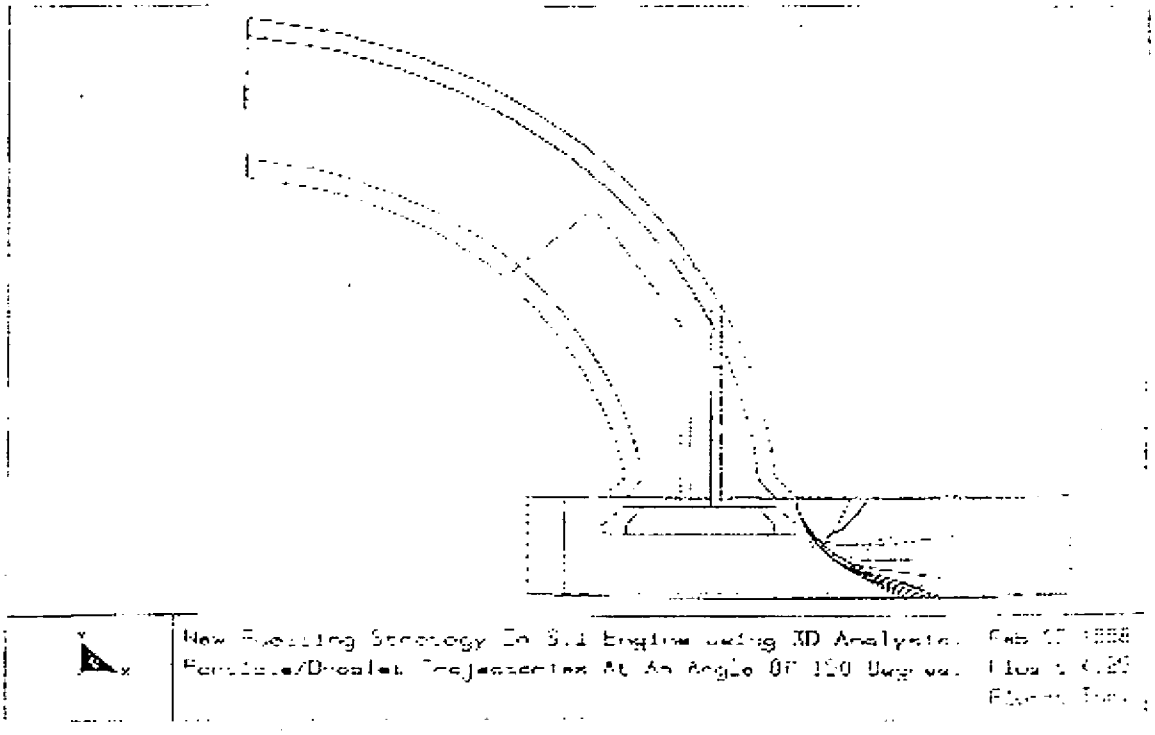
Figure (a) Side view of the droplet trajectories (Injected angle 90 °, engine speed of 3000 rpm).



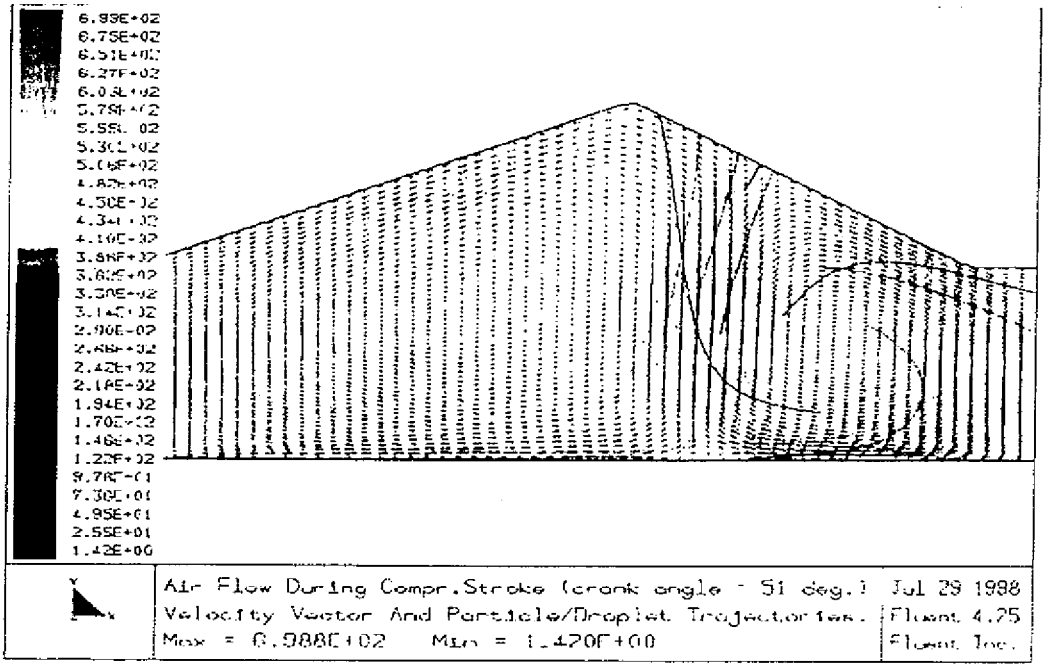
Figure(9)Front view of the droplet trajectories (Injected angle 120 °, engine speed at 3000 rpm).



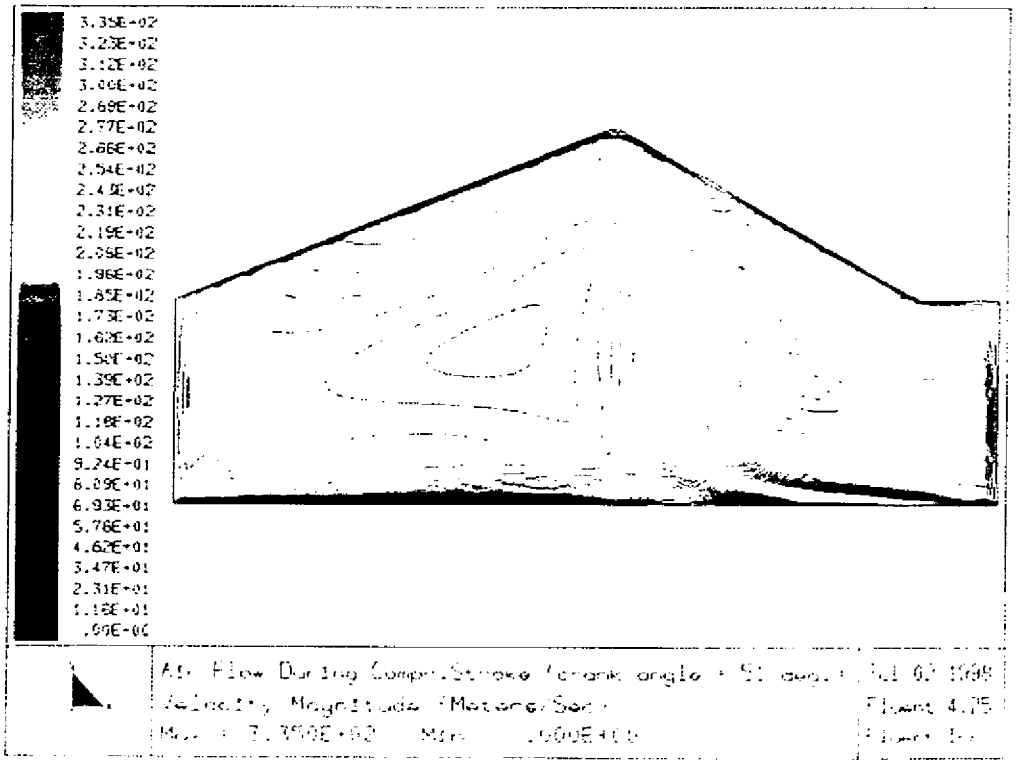
Figure(10)Side view of the droplet trajectories
(Injected angle 120°, engine speed of 3000 rpm).



Figure(11)Front view of the droplet trajectories
(Injected angle 150°, engine speed of 3000 rpm).



Figure(12)Velocity distribution inside the cylinder using normal piston head (flat type).



Figure(13)Velocity magnitude inside the cylinder using normal piston head (flat type)

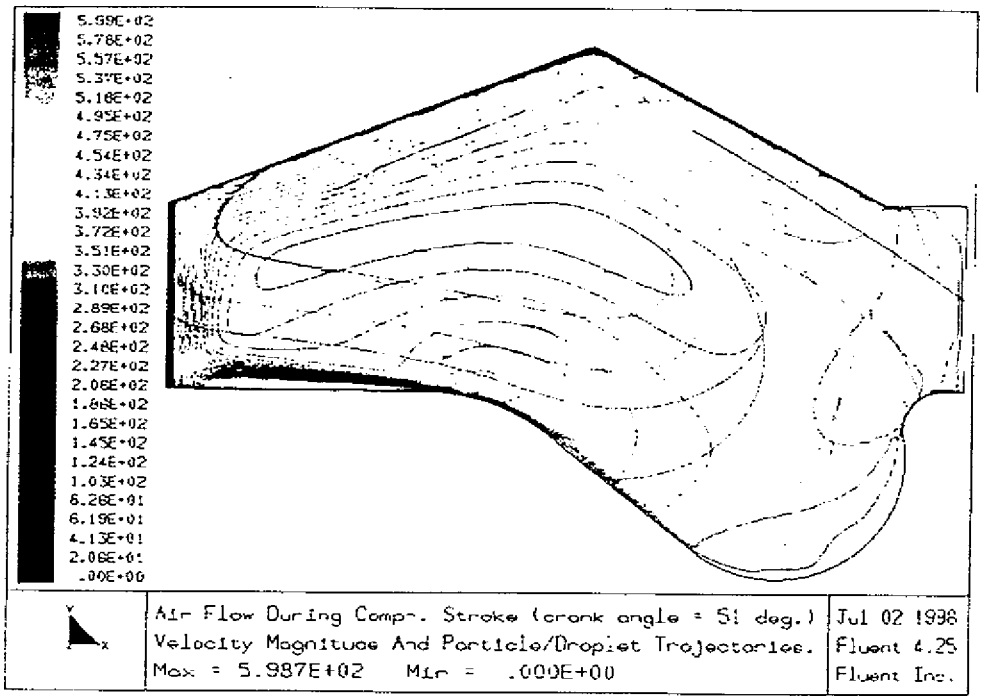


Figure (14) Velocity magnitude inside the cylinder using modified piston head .

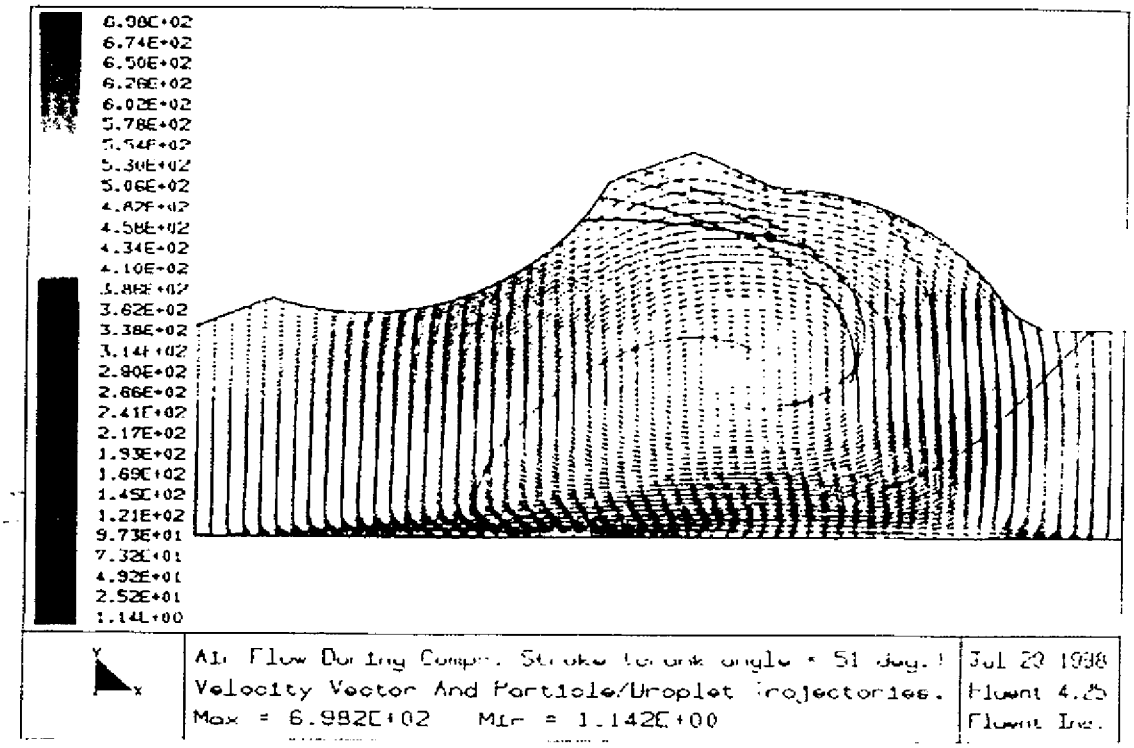


Figure (15) Velocity distribution inside the cylinder using modified combustion chamber.

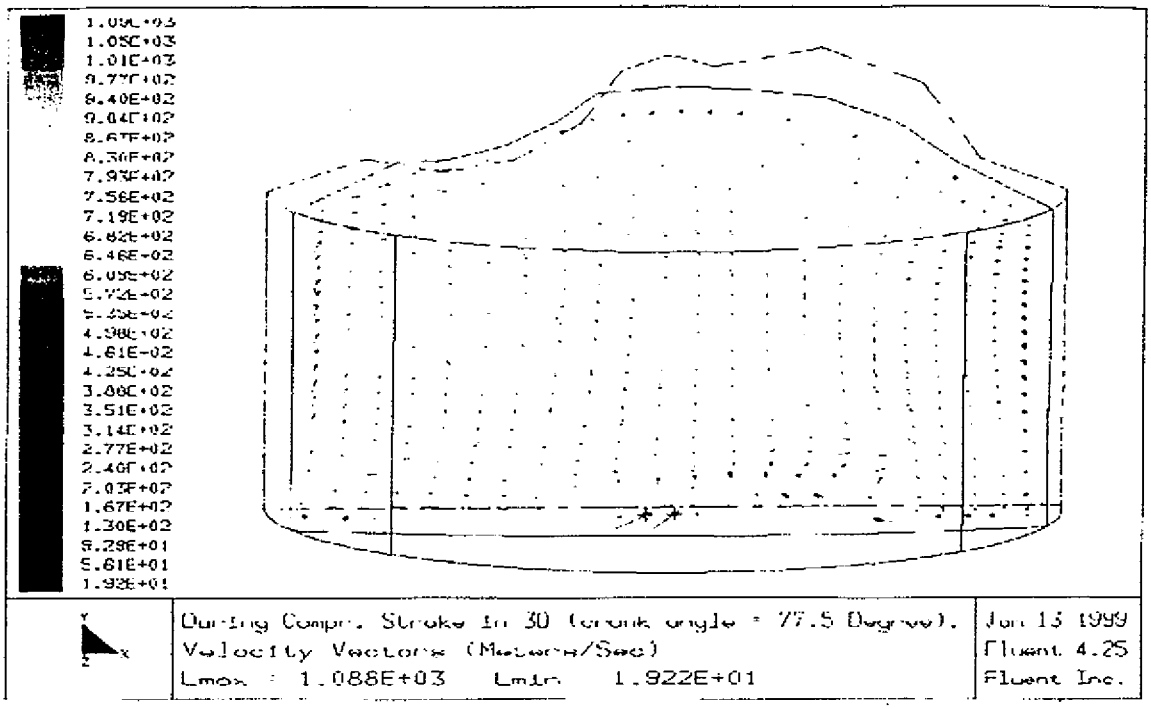


Figure (16) Velocity distribution inside the cylinder with modified combustion chamber (a slice near the center of the cylinder).

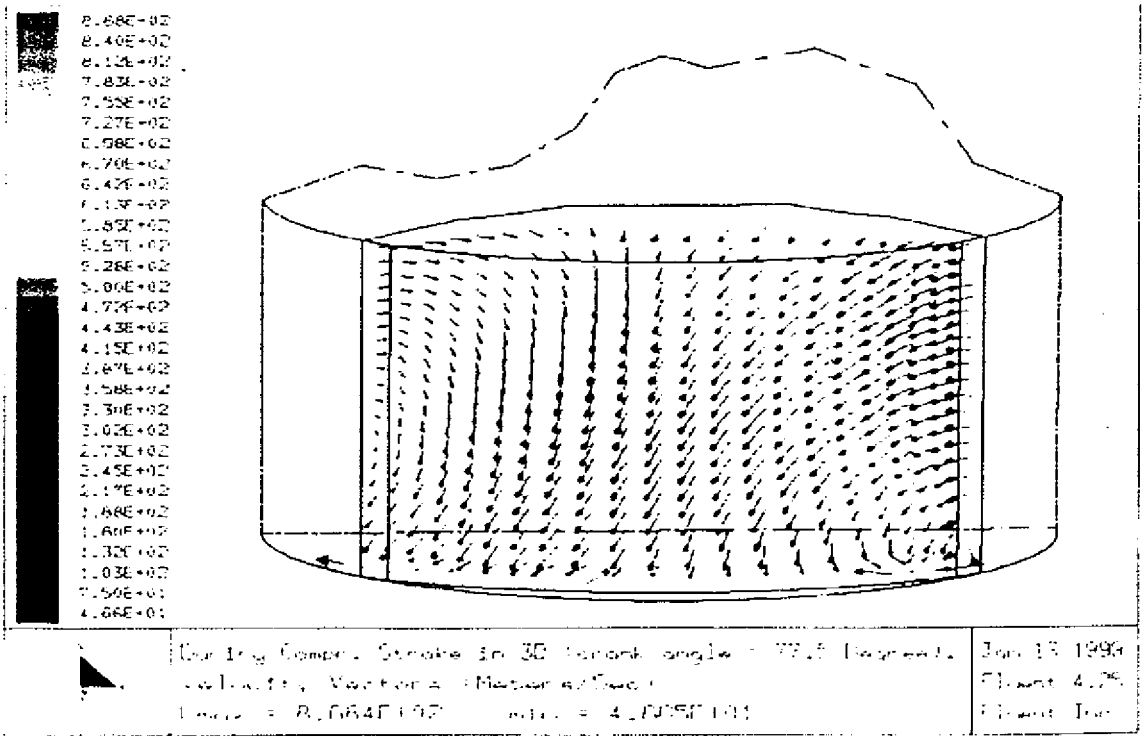


Figure (17) Velocity distribution inside the cylinder with modified combustion chamber (a slice away from the center of the cylinder).