CFD Analysis of Compressible Flow Across A Complex Geometry Venturi

Ilaya Raja[1]
Bharath university

***Abstract*-- A commercial computational fluid dynamics (CFD) package was used to develop a three dimensional, fully turbulent model of the compressible flow across a complex-geometry venturi, such as those typically found in small engine carburetors. The results of the CFD simulations were used to understand the effect of the different obstacles in the flow on the overall discharge coefficient and the static pressure at the tip of the fuel tube. It was found that the obstacles located at the converging nozzle of the venturi do not cause significant pressure losses, while those obstacles that create wakes in the flow, such as the vfuel tube and throttle plate, are responsible for most of the pressure losses. This result indicated that an overall discharge coefficient can be used to correct the mass flow rate, while a localized correction factor can be determined from three-dimensional CFD simulations in order to estimate the static pressure at locations of interest within complex venturis.**

I . INTRODUCTION

1.1 Carburetor

The Carburetor is a device that mixes fuel into the incoming air. The airflow into

the carburetor is controlled by a butterfly valve, and the fuel is added to the mixture though venturi. In a carburetor equipped engine, the air comes into the space for air filter. Air passes through the air filter and then into the carburetor where the fuel is blended with it. Through the intake manifold, it passes and then it is drawn into the cylinders.

1.2 Motorcycle Carburetor Theory

Motorcycle carburetors look very complex, but with a little theory, you can tune your bike for maximum performance. All carburetors work under the basic principle of atmospheric pressure. Atmospheric pressure is a powerful force which exerts pressure on everything. It varies slightly but is generally considered to be 15 pounds per square inch (PSI). This means that atmospheric pressure is pressing on everything at 15 PSI. By varying the atmospheric pressure inside the engine and carburetor, we can change the pressure and make fuel and air flow through the carburetor.

Atmospheric pressure will force high pressure to low pressure. As the piston on a two stroke engine goes up (or goes down on a four stroke engine), a low pressure is formed below the piston inside the crankcase (above the piston on a four stroke). This same low pressure also causes a low pressure inside the carburetor. Since the pressure is higher outside the engine and carburetor, air will rush inside the carburetor and into the engine until the pressure is equalized.

The moving air going through the carburetor will pick up fuel and the fuel will mix with the air.Inside the carburetor there is a venturi, and is a restriction inside the carburetor that forces air to speed up to get through. A river that suddenly narrows can be used to illustrate what happens inside a carburetor. The water in the river speeds up as it gets near the narrowed shores and will get faster if the river narrows even more. The same thing happens inside the carburetor. The air that is speeding up will cause atmospheric pressure to drop inside the carburetor. The faster the air moves, the lower the pressure inside the carburetor. By placing tubes inside the venturi, we can use this lower pressure to put fuel into the air stream.

Most motorcycle carburetor circuits are governed by throttle position and not by engine speed. There are five main metering systems inside most motorcycle carburetors. These metering circuits overlap each other and they are:

- pilot circuit
- throttle valve
- needle jet and jet needle
- main jet
- choke circuit

The pilot circuit has two adjustable parts, the pilot air screw and pilot jet. The air screw can be located either near the back side of the carburetor or near the front of the carburetor. If the screw is located near the back, it regulates how much air enters the circuit. If the screw is turned in, it reduces the amount of air and richens the mixture. If it is turned out, it opens the passage more and allows more air into the circuit which results in a lean mixture. If the screw is located near the front, it regulates fuel. The mixture will be leaner if it is screwed in and richer if screwed out. If the air screw has to be turned more than 2 turns out for best idling and performance, the next smaller or bigger size pilot jet will be needed.

The pilot jet is the part which supplies most of the fuel at low throttle openings. It has a small hole in it which restricts fuel flow through it. Both the pilot air screw and pilot jet affects carburetion from idle to around 1/4 throttle. The slide valve affects carburetion between 1/8 thru 1/2 throttle. It especially affects it between 1/8 and 1/4 and has a lesser affect up to 1/2. The slides come in various sizes and the size is determined by how much is cut away from the backside of it, The larger the cutaway, the leaner the mixture (since more air is allowed through it) and the smaller the cutaway, the richer the mixture will be. Throttle valves have numbers on them that explains how much the cutaway is. If there is a 3 stamped on the slide, it has a 3.0mm cutaway, while a 1 will have a 1.0mm cutaway (which will be richer than a 3). The slide valve affects how much air can pass through the slide from around 1/8 thru 1/4 throttle. The jet needle and needle jet affects carburetion from 1/4 thru 3/4 throttle. The jet needle is a long tapered rod that controls how much fuel can be drawn into the carburetor venturi. The thinner the taper, the richer the mixture. The thicker the taper, the leaner the mixture since the thicker taper will not allow as much fuel into the venturi as a leaner one. The tapers are designed very precisely to give different mixtures at different throttle openings. Jet needles have grooves cut into the top. A clip goes into one of these grooves and holds it from falling or moving from the slide. The clip position can be changed to make an engine run richer or leaner. If the engine needs to run leaner, the clip would be moved higher.

This will drop the needle farther down into the needle jet and cause less fuel to flow past it. If the clip is lowered, the jet needle is raised and the mixtures will be richer. The clip position changes the air/fuel mixture on the jet needle. The needle jet is where the jet needle slides into. Depending on the inside diameter of the needle jet, that will affect the jet needle. The needle jet and jet needle work together to control the fuel flow between the 1/8 thru 3/4 range. Most of the tuning for this range is done to the jet needle, and not the needle jet.

The main jet controls fuel flow from 3/4 thru full throttle. Once the throttle is opened far enough, the jet needle is pulled high enough out of the needle jet and the size of the hole in the main jet begins to regulate fuel flow. Main jets have different size holes in them and the bigger the hole, the more fuel that will flow (and the richer the mixture). Higher numbers on main jets will have a richer air/fuel mixture than smaller holes. Hence, the main jet controls fuel flow from 3/4 thru full throttle. The choke system is used to start cold engines. Since the fuel in a cold engine is sticking to the cylinder walls because of condensation, the mixture is too lean for the engine to start. The choke system will add fuel to the engine to compensate for the fuel that is stuck to the cylinder walls. Once the engine is warmed up, condensation is not a problem, and the choke is not needed. The air/fuel mixture must be changed to meet the demands and needs of the engine. The ideal air/ fuel ratio is 14.7 grams of air to 1 gram of fuel. This ideal ratio is only achieved for a very short period while the engine is running. Due to the incomplete vaporization of fuel at slow speeds or the additional fuel required at high speeds, the actual operational air/fuel ratio is usually richer. Figure 1.6 shows the actual air/fuel ratio for any given throttle opening.

II. BASICS OF COMPUTATIONAL FLUID DYNAMICS

CFD provides numerical approximation to the equations that govern fluid motion. Application of the CFD to analyze a fluid problem requires the following steps. First, the mathematical equations describing the fluid flow are written. These are usually a set of partial differential equations. These equations are then discretized to produce a numerical analogue of the equations. The domain is then divided into small grids or elements. Finally, the initial conditions and the boundary conditions of the specific problem are used to solve these equations. The solution method can be direct or iterative. In addition, certain control parameters are used to control the convergence, stability, and accuracy of the method.

All CFD codes contain three main elements: (1) A pre-processor, which is used to input the problem geometry, generate the grid, and define the flow parameter and the boundary conditions to the code. (2) A flow solver, which is used to solve the governing equations of the flow subject to the conditions provided. There are four different methods used as a flow solver: (i) finite difference method; (ii) finite element method, (iii) finite volume method, and (iv) spectral method. (3) A post-processor, which is used to massage the data and show the results in graphical and easy to read format.

III. CFD TOOL – ANSYS CFX

CFX is a commercial Computational Fluid Dynamics (CFD) program, used to simulate fluid flow in a variety of applications. The ANSYS CFX product allows engineers to test systems in a virtual environment. The scalable program has been applied to the simulation of water flowing past ship hulls, gas turbine engines (including the compressors, combustion chamber, turbines and afterburners), aircraft aerodynamics, pumps, fans, HVAC systems, mixing vessels, hydro cyclones, vacuum cleaners etc.

ANSYS CFX is a high-performance, general purpose CFD program that has been applied to solve wide-ranging fluid flow problems for over 20 years. At the heart of ANSYS CFX is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly.

The modern, highly parallelized solver is the foundation for an abundant choice of physical models to capture virtually any type of phenomena related to fluid flow: laminar to turbulent (including transition), incompressible to fully compressible, subsonic to trans- and supersonic, isothermal or with heat transfer by convection and/or radiation, non-reacting to combusting, stationary and/or rotating devices, single fluids and mixtures of fluids in one or more phases (incl. free surfaces), and much, much more.

The solver and its many physical models are wrapped in a modern, intuitive, and flexible GUI and user environment, with extensive capabilities for customization and automation using session files, scripting, and a powerful expression language.

But ANSYS CFX is more than 'just' a powerful CFD code: with its integration into the ANSYS Workbench platform, you benefit from superior bi-directional connections to all major CAD systems, powerful geometry modification and creation with ANSYS Design Modeler, advanced meshing technologies in ANSYS Meshing, and easy drag-and-drop transfer of data and results to share between applications (e.g. to use a fluid flow solution in the definition of a boundary load of a subsequent structural mechanics simulation).

IV. SCOPE AND OBJECTIVE OF THE PROJECT

Large volumes of small engines (two wheelers) are being sold in India every year. Its emissions comprise a significant percentage of total pollutants in India. Better understanding of carburetor performance and modeling could lead to better fuel mixture control and lower emissions from small engines.

Carburetors are mostly being produced by small scale industries. They must be durable, light-weight, inexpensive, fuel-efficient and clean. Constraints such as price, low-weight and small-packaging have made it difficult for manufacturers to use technological solutions widely used in automotive engines like electronic control, fuel injection, exhaust gas reticulation and exhaust after-treatment. Also, present Pollution norms dictating an improved carburetor with more precise intake mixture control. In order to improve the performance, major modification in the design of carburetor cannot be afforded by such industries.

Hence Carburetor will be large player of future small engines. In the recent days many sophisticated R&D tools have come up to study and improve performance of such equipments. This has motivated me to take this as opportunity to analyse the complicated carburetor venturi passages in order to improve the performance.

The project involves flow of compressible fluid through a confined passage and process is to be optimized on basis of flow behavior of fluid through system.

Computational Fluid Dynamics (CFD) is capable of simulating any physical flow process. And hence it helps better understanding of the flow pattern and to study all aspects of the details of flow field, turbulence, recirculation zones etc. Computational Fluid Dynamics is considered to be the most effective tool for flow analysis of carburetor venturi. There is a lot of scope to apply CFD for this problem.

The objective of the project is as follows

- To carry out three dimensional CFD analysis of carburetor venturi to understand the effect of the various obstacles present in the flow domain like the fuel-tube, throttle plate and to optimize the design of carburetor by carrying out geometrical changes based on results obtained from CFD analysis of existing model.
- To perform CFD analysis by considering the following models.
  - a) Ideal carburetor venturi
  - b) Existing carburetor venturi
  - c) Modified carburetor venture

A real carburetor venturi has details in its geometry that create disturbances in the flow, and may cause pressure losses that cause deviations from an ideal isentropic flow. Examples of these carburetor parts are the choke plate, the throttle plate, the fuel tube, side passages to secondary systems and, sometimes, an additional concentric fuel tube in the venturi throat. Some details of typical carburetors used in small engines are shown in Figure.1

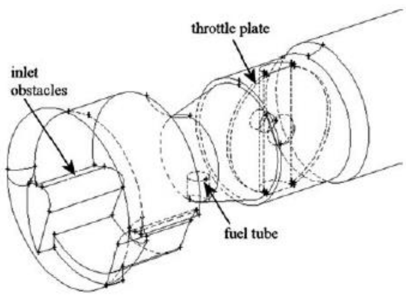

Fig-1 Carburetor Model

The pressure losses created by these elements reduce the mass flow rate that could be driven through the venturi for a given pressure difference between the inlet of the venturi and the intake manifold.

In the present study, the fuel tube and the throttle plate were modeled with CFX, in order to gain a better understanding of the characteristics of the flow, and how it is affected by these parts.

V .RESULT AND DISCUSSION

5.1Ideal carburetor venturi

Figure 2 shows static pressure, velocity turbulent kinetic energy and total pressure across the venture without fuel tube. Except at the throat after venture the pressure variation is almost constant (no pressure fluctuation).The velocity field shows that velocity is almost constant behind the venturi. The turbulence kinetic energy field shows that the intensity of turbulence created. The highest turbulence region created at near wall throat. The total pressure (stagnation pressure) shows that it is uniform throughout the flow. Generally the reduction in stagnation pressure creates wake region (turbulence region).

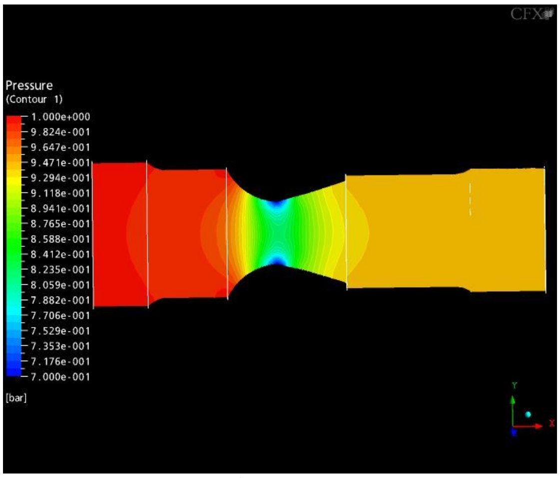

Fig 2 a) Static Pressure

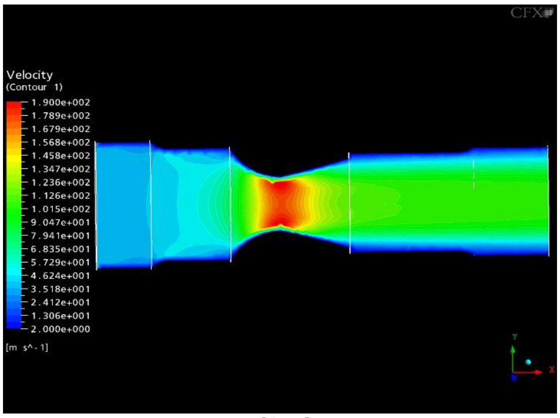

Fig 2 .b)Velocity

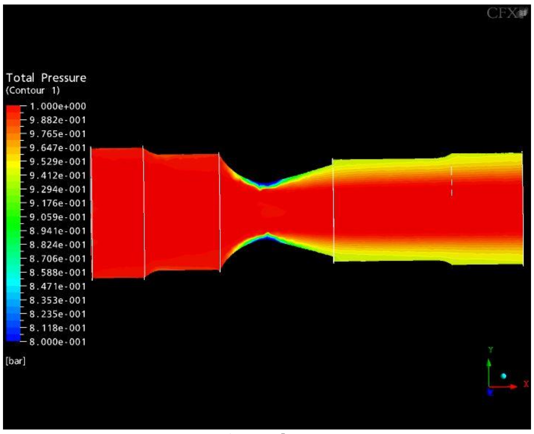

Fig 2 c) Total Pressure

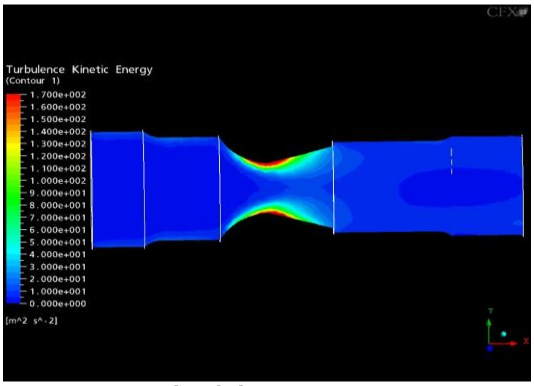

Fig 2 d)Turbulence Kinetic energy

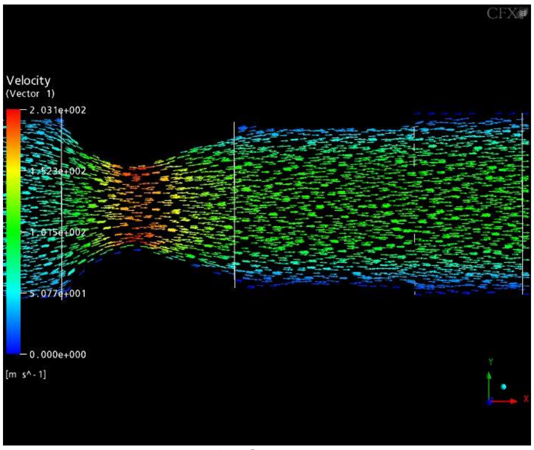

Fig 2 e)Velocity Vectors

2.Steady airflow across carburetor venturi without obstacles

5.2 Modified Design of Throttle Plate

A throttle plate was modeled with its body divided in two identical half-plates with individual screws for them as shown in Figure. They were located at the same downstream location from the venturi throat as the original throttle plate.

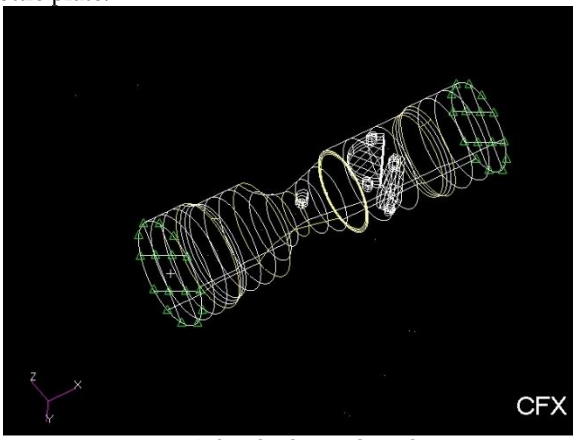

Fig. 3a)Throttle Plate angle 60 deg

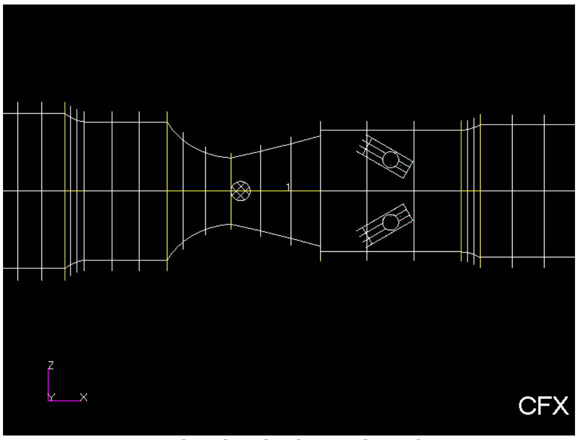

Fig 3.b) Throttle Plate angle 60 deg

5.3 Carburetor venturi with two throttle at 60 degrees

The models were analyzed for the same boundary conditions. Fig 3 shows the parameter variation for two throttle for 60 deg opening position. The pressure has got decreased at throat and the pressure uniformity has been observed at downstream of venturi. The velocity and stagnation pressure variation shows that uniformity has maintained along the on either side with respect to the axis. The reduced kinetic energy variation has been observed beyond the throttle plate which will consequently reduce the fluctuation in the flow and hence the less energy loss.

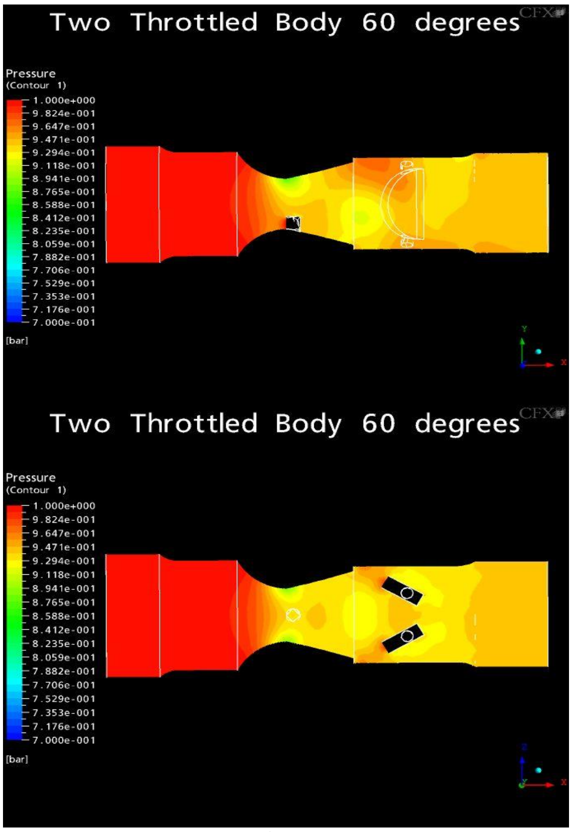

Fig 4.a)Pressure Variation

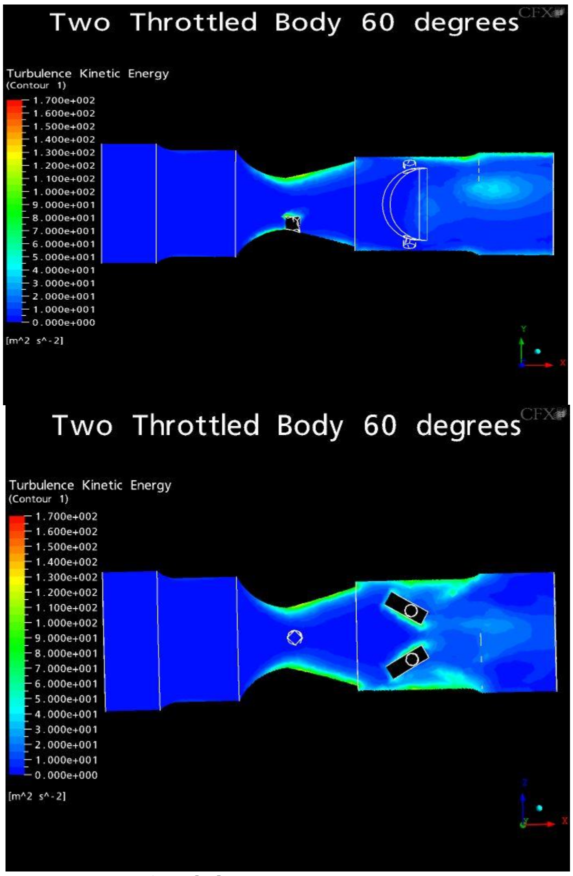

Fig 4 a) Turbulence Kinetic energy Variation

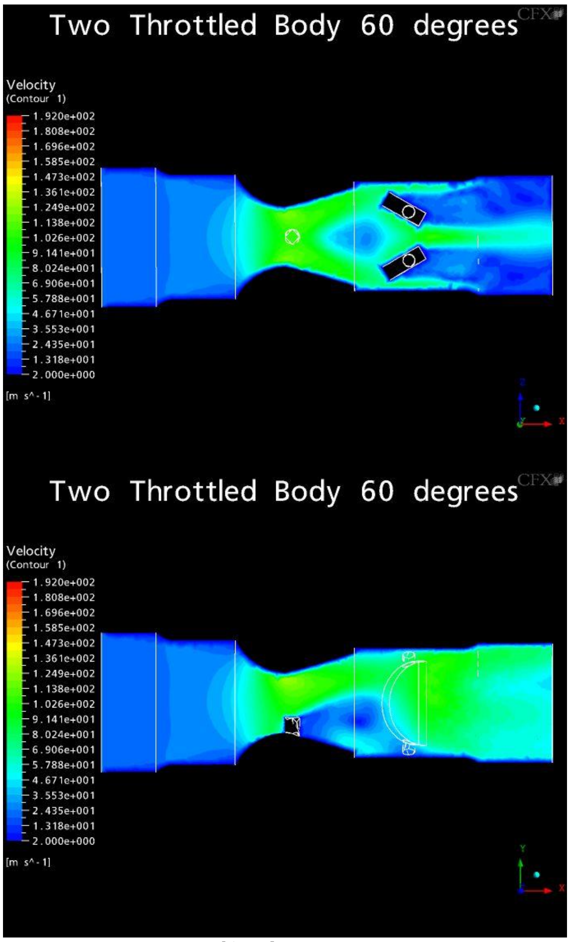

Fig 4 . b) Velocity Variation

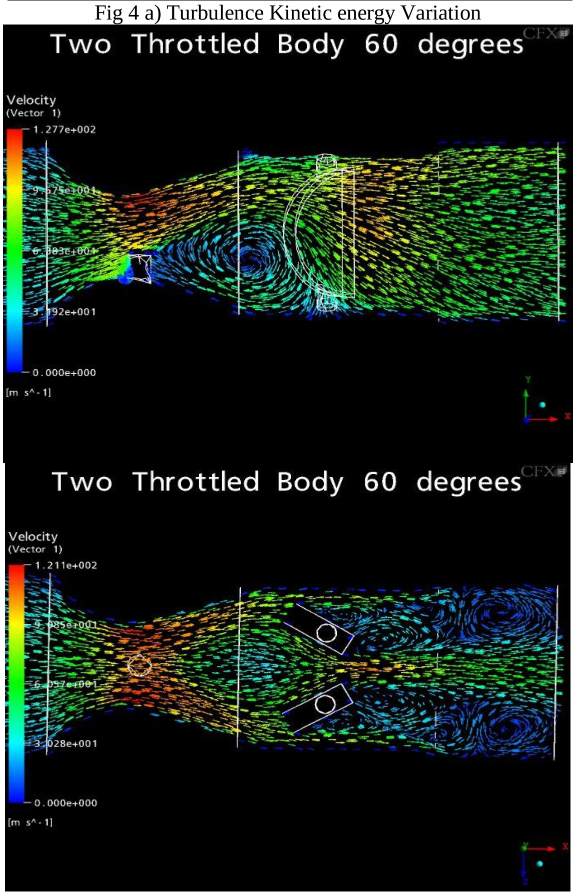

Fig 4 b) Velocity Vectors

Fig 4 Carburetor venturi with Two throttle at 60

VI. CONCLUSION

CFD analysis has been carried out using commercial CFD solver CFX to analyze the flow behavior of the existing carburetor body used in small engines.

The result of conventional throttle positions indicates that flow recirculation at downstream which causes pressure fluctuations and increased stagnation pressure loss which is undesirable. More over the velocity vectors for various throttle plate positions also show that the recirculation in the flow just before throttle plate (front views).

The modified model also shows comparatively increased volumetric efficiency. The analyses of the modified model showed that the design achieves more symmetric and organized flow at the downstream of carburetor. This simple design change has the potential for improving mixture distribution downstream of the carburetor without major changes in the carburetor design

REFERENCE

[1] Y. Asano, T. Chuma, H. Haga, and T. Mochida. Effects of air-bleed systems on carburetor performance. Bulletin of JSME, pages 691–698, 1988.
[2] M. Ehara, Y. Kinabara, K. Shinoda, T. Meguro, and T. Ishihara. A study of carburetor design. SAE technical report, 1990.
[3] M. Furuyama. Fuel supply characteristics of idle-system of carburetor under steady state conditions. Bulletin of JSME, pages 722–729, 1988.
[4] M. Furuyama. Fuel supply characteristics in the transition region of fixed-venturi type carburetor. Bulletin of JSME, pages 842–848, 1991.
[5] I. Isobe and Y. Asano. Discharge beginning points of fuel and bleed-air of carburetor The University of Chiba Report, pages 173–178, 1998
[6] D. K. Jagdish, S. Govindarajan, V. Balasubramaniam, and V. Ganesan. Theoretical and experimental investigation of metering characteristics of a variable venturi mechanically controlled carburetor. SAE technical report, 2001.
[7] B. Sendyka and J. Filipczyk. Simulation of the characteristic of a carburetor of an internal combustion engine. SAE technical report, 1995.
[8] D. L. Harrington and J. A. Bolt. Analysis and digital simulation of carburetor metering SAE technical report, 1990.
[9] B. Sendyka and J. Filipczyk. Simulation of the characteristic of a carburetor of an internal combustion engine. SAE technical report, 1995.
[10] B. Sendyka and W. Heydel. The analysis of a constant depression carburetor with a fuel feeder. SAE technical report, 1994.
[11] M. Furuyama and H. Ohgane. A comparison of pulsating and steady flows in terms of carburetor characteristics. JSAE Review, pages 18–23, 1987.
[12] P. J. Moss. Pulsation enrichment in carburetors. Automotive Engineer, pages 53–56, 1980.
[13] B. Wu, Y. Feng, and J. Liu. Numerical simulation of gas-liquid two-phase flow in motorcycle carburetor. In Proceedings of the International Symposium on Multiphase Flow, pages 271–275, 1997.
[14] Diego Arias, Timothy A. Shedd. CFD analysis of compressible flow across complex geometry venturi – Journal of Fluids Engineering – Vol.129 pages1193-1202, 2007