(IJAER) 2015, Vol. No. 10, Issue No. V, November **e-ISSN: 2231-5152/ p-ISSN: 2454-1796**

DESIGN AND ANALYSIS OF MODIFIED CARBURETTOR USING CFD

***S.Palanisamy, *S.Ramanathan, **R. Naveenkumar**

**Assistant Professor, Department of Mechanical Engineering, CARE Group of Institutions, Trichy, Tamilnadu, India*

***Senior Lecturer, Department of Mechanical Engineering, CARE Group of Institutions, Trichy, Tamilnadu, India.*

ABSTRACT

In this paper, nozzle and throttle plate was modeled and analyzed in order to have better understanding of the flow in complex venturi. The carburettor body has been remodeled with two throttle bodies replacing conventional throttle. Analysis have been performed to study flow field with modified design. Further a comprehensive comparative study was made with the existing and modified design. A commercial computational fluid dynamics package were used to develop a three dimensional, fully turbulent model of the compressible flow across a complex geometry venturi, such as those found in low speed engine carburettors. The results of CFD simulation were utilized to understand the effect of different obstacles in flow on the mass flow rate and static pressure at the tip of fuel Jet. The results indicated that the efficiency of the engine has been increased by the modified carburettor.

Keywords: *CFD, Carburettor and Throttle.*

1. INTRODUCTION

1.1. GENERAL

Carburettors are mechanical devices that are used in gasoline / petrol engines in order to deliver the air-fuel mixture in the right proportion to the engine under all running conditions. This process is called carburetion. The carburettor achieves this by means of a complex set of passages and the flow varies from laminar to fully turbulent, two-phase flow, pulsating flow etc due to rapid changes in the operating conditions and changes in pressure and temperature.

1.2 LITERARURE REVIEW

Studies have been conducted to predict fuel flow versus air flow for a carburettor with an air bleed system. The effects of different carburettor parts on fuel flow and air-fuel ratio can be seen in this kind of studies. In this the main fuel orifice is the part responsible for the main trend of the air fuel ratio under moderate-to-high air flow; the idle system is responsible for fuel delivery under low air flow; and the air bleed system is responsible for increasing fuel flow during the transition between the idle system and the main fuel system. All of these studies had as a limitation the computational capabilities at the time they were performed. A significant effort was required in order to solve the nonlinear system of equations that represented the flow network. For example, in references [3] and [4], Furuyama developed the equations for a carburettor with idle and main circuits, and by mathematical substitution simplified the theoretical model until a single equation was obtained. This strategy results in a loss of information about intermediate variables, like static pressure at network nodes and flows across secondary passages. Harrington [8] used the steady flow assumption to predict the fuel flow for a two-barrel carburettor used in an eight-cylinder engine. The results agreed well with experiments. Having a large number of cylinders, the assumption of steady air flow seemed to be reasonable.

In single- and two-cylinder engines, the airflow at the venturi is expected to be pulsating, and the application of a steady state model is expected to be limited. However, D.K.Jagdish [6] used a steady flow assumption for a single-cylinder engine and claimed good agreement. Sendyka's results [9, 10] showed that instantaneous and integrated air-fuel ratios that were leaner than those seen in real engines. It was thought that the difference between the model and experiments was caused by the inability to capture dynamic effects of the fuel flow.

Experimental studies performed by Furuyama and Ohgane [11] and Moss [12] showed that the pulsating nature of the air flow affects the amount of fuel delivered by the carburettor.

Furuyama found that the effect of pulsating air flow on fuel flow may be classified as: i) when the throttle plate opening is large and air flow is low, the fuel flow is higher at pulsating flow than at steady flow, and ii) when the throttle plate opening is large and air flow is high, the fuel flow is lower at pulsating flow than at steady flow. Moss' experiments [12] agreed with the conclusions for the first case. Both researchers proposed that the fuel flow under dynamic air flow may be calculated by using the steady state prediction, and then corrected with a pulsation-correction factor. Two special considerations must be taken when predicting the fuel flow from the carburettor circuits: the characterization of the two-phase flow inside the emulsion tube and the characterization of the small metering orifices.

The only known work that has used CFD for the characterization of the flow across the carburettor was done by Wu, Feng and Liu [13]. But in their work, the carburettor was represented as a two-dimensional channel where the fuel tube was a large obstacle in the flow field. The only results shown in this work are the static pressure drop along the axis of the carburettor.

2. DEFINITION OF PROBLEM AND OBJECTIVE

A real carburettor 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 carburettor 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.

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.

More over increase in volumetric efficiency is essential since engines with higher volumetric efficiency will generally be able to run at higher speeds and produce more overall power due to less parasitic power loss moving air in and out of the engine.

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.

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 carburettor performance and modeling could lead to better fuel mixture control and lower emissions from small engines.

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 carburettor 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 carburettor 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 carburettor 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 carburettor venturi
  b) Existing carburettor venturi

C) Modified carburettor venturi

2.1 DIMENSIONS OF MODEL

Inlet diameter of venturi	= 25 mm
Throat diameter	= 12 mm
Outlet diameter of venturi	= 20 mm
Throttle plate distance from throat	= 25mm
Distance between inlet and throat	= 37 mm
Distance between throat and outlet	= 54 mm
Length of venturi	= 24 mm
Length of carburettor	= 91 mm

2.2 CONSIDERATIONS IN CFD

The most fundamental consideration in CFD is how one treats a continuous fluid in a discretised fashion on a computer. One method is to discretise the spatial domain into small cells to form a volume mesh or grid, and then apply a suitable algorithm to solve the equations of motion (Euler equations for inviscid and Navier–Stokes equations for viscous flow).

This is necessitated by the fact that computational methods can only be applied to finite number of elements and not to a continuous domain as this would translate to practically infinite nodes for computation and thus put a tremendous load on the processing hardware and render the solution impossible.

In addition, such a mesh can be either irregular (for instance consisting of triangles in 2D, or pyramidal solids in 3D) or regular; the distinguishing characteristic of the former is that each cell must be stored separately in memory. Where shocks or discontinuities are present, high resolution schemes such as Total Variation Diminishing (TVD), Flux Corrected Transport (FCT), Essentially Non-Oscillatory (ENO), or MUSCL schemes are needed to avoid spurious oscillations (Gibbs phenomenon) in the solution.

2.3 MESHING

Mesh generation is the process by which spatial discretization of CFD model is accomplished. Meshing is based on tetrahedron element discretization. Surface and volume meshes were generated in CFX- Build module by defining the type of meshing element and mesh element size.

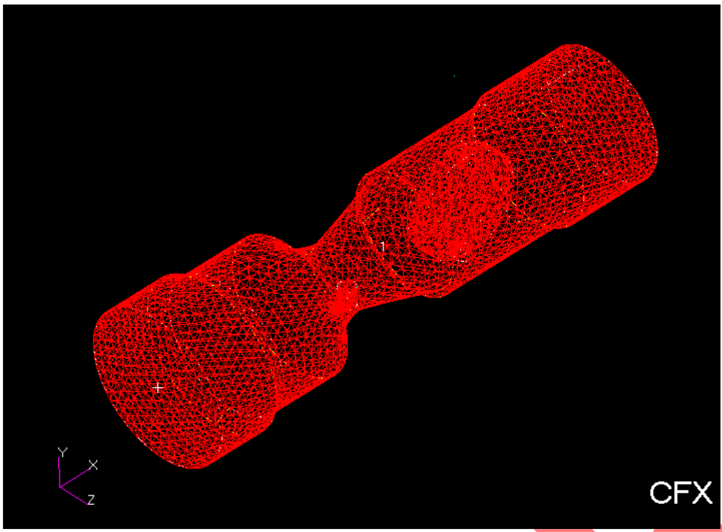

Fig 2.1 Meshed model -90° throttle opening position

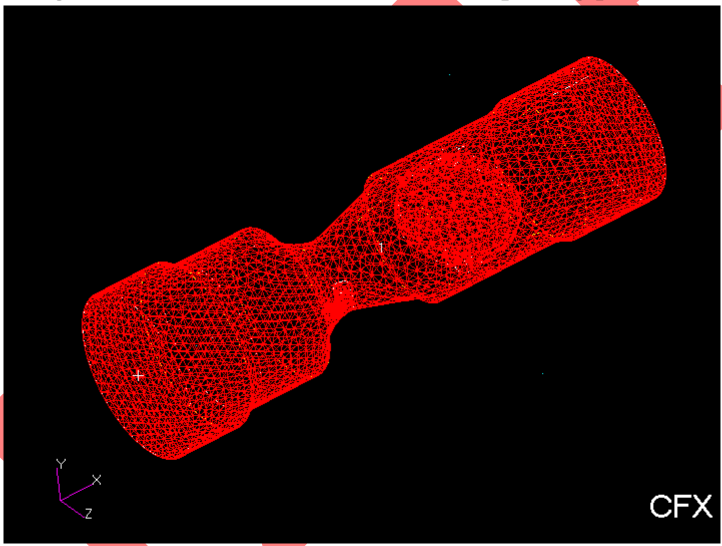

Fig 2.2 Meshed model -45° throttle opening position

Mesh element size

Element type : Tetrahedron
Maximum Edge length : 1.5 mm
Mesh Type : Volume mesh
No. of Elements : 112000

2.4 BOUNDARY CONDITIONS AND DOMAIN SPECIFICATIONS

Inlet Boundary Conditions

Temperature = 293 K
Pressure = 1 bar

Outlet Boundary Condition

Pressure = 0.9 bar

Wall boundary Conditions

Wall influence on flow : No slip

Wall Roughness : Smooth wall
Heat transfer : Adiabatic

Domain Specification

Domain Type : Fluid Domain
Fluid : air
Heat transfer Model : Thermal Energy
Turbulence Model : K-Epsilon

3. RESULTS AND DISCUSSION

3.1 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.

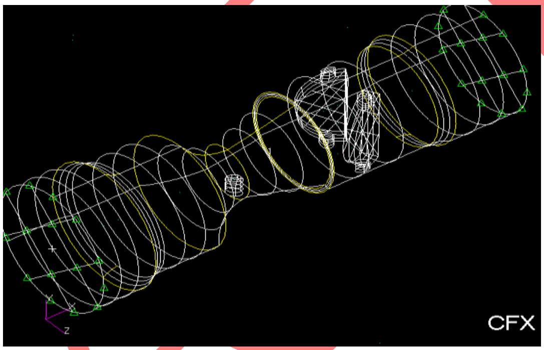

Fig 2.3 Carburettor venturi with Double throttle Body

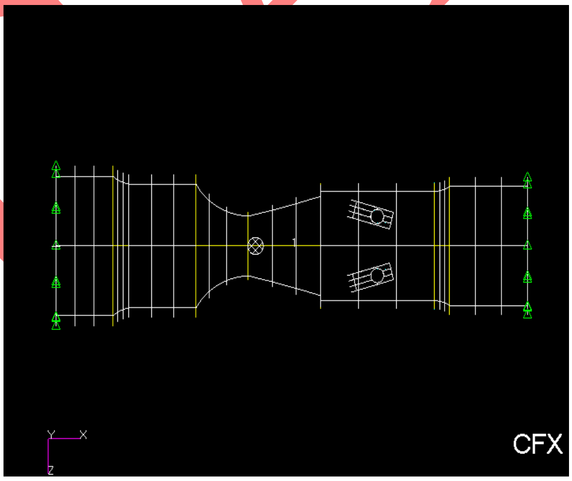

Fig 2.4 Carburettor venturi with Double throttle plate angle 75°

(IJAER) 2015, Vol. No. 10, Issue No. V, November **e-ISSN: 2231-5152/ p-ISSN: 2454-1796**

3.2 Carburettor venturi with double throttle at 75 degrees

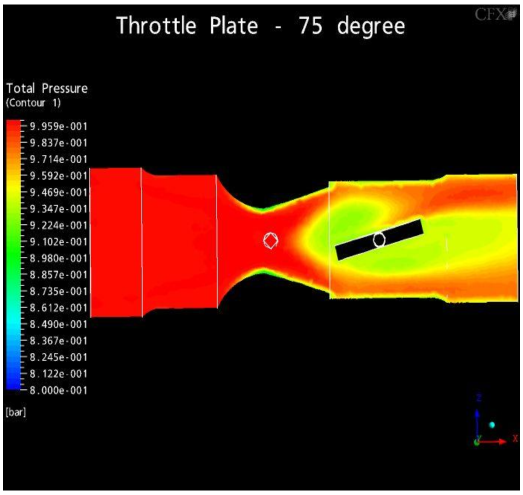

Fig 3.1 Total Pressure

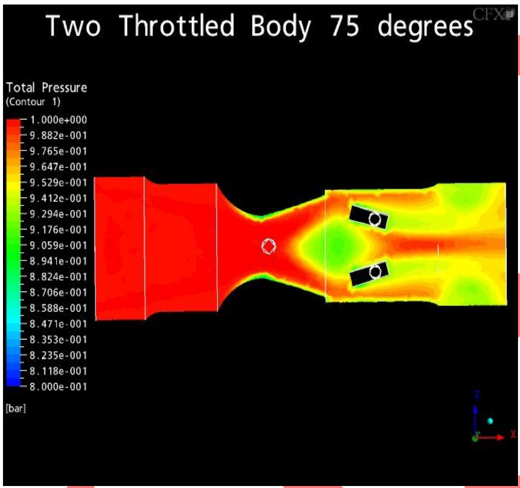

Fig 3.2 Total Pressure

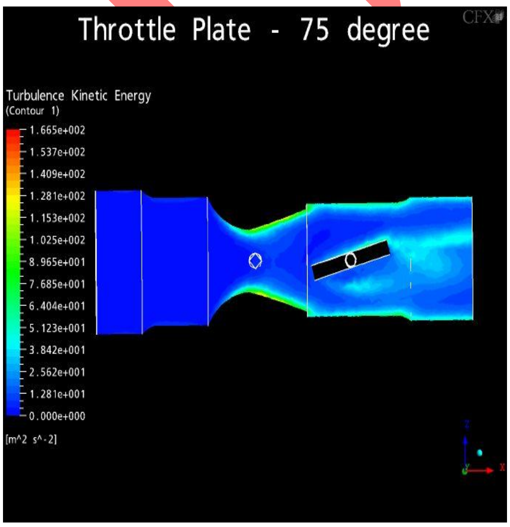

Fig 3.3 Turbulence Kinetic energy

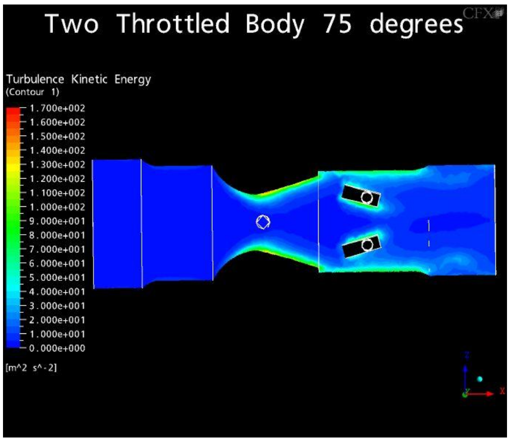

Fig 3.4 Turbulence Kinetic energy

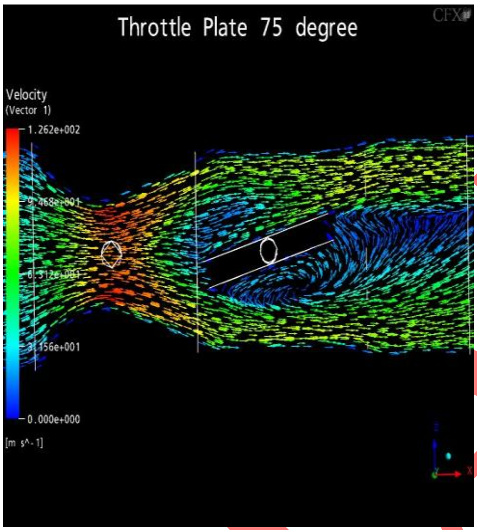

Fig 3.5 Velocity Vectors

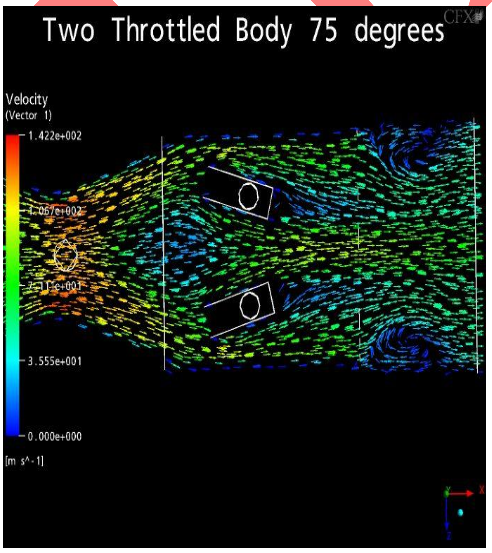

Fig 3.6 Velocity Vectors

The models were analyzed for the same boundary conditions. The analyses of results for 75 degrees show that reduced stagnation pressure loss at downstream. The kinetic energy field shows that it is almost constant throughout the flow. The velocity vectors clearly show that reduced flow recirculation at downstream

3.3 STUDY OF MASS-FLOW RATE

Mass-flow rate is simply the mass of a substance that flows through a surface per unit time. In this project, the mass-flow rate of air was recorded for all the nine carburettor configurations using the ANSYS CFX function definition tool.

Compared to the mass-flow rate through the venturi with no obstacles, the introduction of the fuel tube reduces the mass flow rate by 69.7 % in the case of the 3mm fuel tube and by 56.2 % in the case of the 6mm fuel tube. This shows that the 6mm fuel tube has a more pronounced deleterious effect than the 3mm fuel tube.

The introduction of the throttle plate causes a further dip in the mass-flow rate of air. The percentage reductions have been shown in the graph below. An interesting trend that is observed in this study is that comparatively, the split throttle plate configurations show a relative increase in the mass flow rate for the same angular position. This means that, for the same available area for air flow, the split throttle plates allow more air to flow through the carburettor. This can be attributed to the symmetrical flow and the reduction in the undesirable eddy flow formations effected by the new designs.

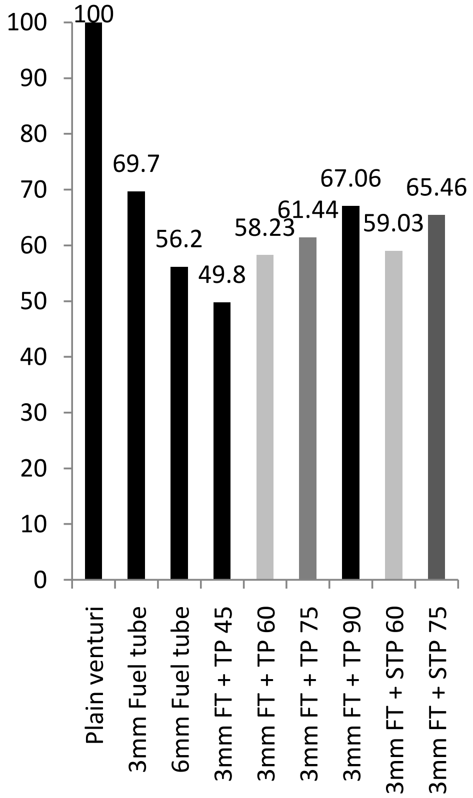

Fig 3.7 Relative percentages of mass-flow rate

From the graph, it can be seen that in terms of mass flow rate the split throttle plate design offers the following theoretical advantages quantitatively:

At 60 degrees, the proposed design offers a marginal increase of **0.8 %.**

At 75 degrees, the proposed design offers an increase of **4.02 %.**

Thus, it proves that the proposed design offers a marginal advantage at low speeds and a much more substantial advantage at higher speeds.

This is a favourable observation as the mass flow rate is directly proportional to the volumetric efficiency of an engine. An increase in the mass flow rate would have a similar incremental effect in the performance of the engine.

3.4 RESULTS OF QUANTITATIVE ANALYSIS

Description	Pressure drop at throat %	Kinetic energy drop at throat %
Single Throttle 45deg	77.49	33.17
Single Throttle 60deg	79.7	41.54
Single Throttle 75deg	81.18	47.02
Single Throttle 90deg	80.38	43.958
Doublethrottle 60	80.23	43.318
Doublethrottle 75	82.61	51.652

Table 3.1 Quantitative Analysis

3.4 VOLUMETRIC EFFICIENCY

The volumetric efficiency for the various throttle opening positions has been calculated by using the following relation

$m_a = \Delta P\, V_D\, ή_{vol}\, N / R\, T$

where,

m_a – mass flow rate in kg/sec

ΔP– Change in pressure in N/m^2

V_D – Displacement Volume in m^3

N – Engine Speed in rpm

$ή_{vol}$ – Volumetric efficiency

R – Gas Constant J/kg K

T – Inlet Temperature in K

Engine Specification

Make : TVS XL Super
Type : 2 Stroke
Displacement : 69.9 CC
Max. Power : 2.61 KW – 7000 rpm
Max. Torque : 5 N-m
Bore Diameter : 46 mm
Stroke Length : 42 mm

$V_D = \pi /4 * D^2 * L$

$= \pi /4 *(46 \times 10^{-3})^2 * 42 \times 10^{-3}$

$= 6.979 \times 10^{-5}\ m^3$

Calculation of volumetric Efficiency -Throttle Angle 75°

$m_a = \Delta P\ V_D\ \acute{\eta}_{vol}\ N /R\ T$

$m_a = 0.01534$ kg/sec

$T = 292.35$ K

$\Delta P = (0.99773 – 0.94398) \times 10^5$

$\Delta P = 5339\ N/m^2$

$0.01534 = 5339 * 6.979 \times 10^{-5} * \acute{\eta}_{vol} * 7000 /287*292.35$

$\acute{\eta}_{vol} = 0.4935 = 49.35\%$

S.No	Description	Volumetric efficiency
1	Ideal carburettor venturi	95.63%
2	Throttle angle 60 deg	45.61%
3	Throttle angle 75 deg	49.35%
4	Throttle angle 90 deg	46.03%
5	Double Throttle angle 60 deg	47.39%
6	Double Throttle angle 75 deg	53.96%

Table 3.2 Volumetric efficiency

CONCLUSION

CFD analysis was done using commercial CFD solver CFX to analyze the flow behavior of the existing carburettor 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.

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 carburettor. This simple design change has the potential for improving mixture distribution downstream of the carburettor without major changes in the carburettor design

REFERENCES

[1] Y. Asano, T. Chuma, H. Haga, and T. Mochida, Effects of air-bleed systems on carburettor performance. Bulletin of JSME, pages 691–698, 1988.

[2] M. Ehara, Y. Kinabara, K. Shinoda, T. Meguro, and T. Ishihara, A study of carburettor design. SAE technical report, 1990.

[3] M. Furuyama, Fuel supply characteristics of idle-system of carburettor 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 carburettor. Bulletin of JSME, pages 842–848, 1991.

[5] I. Isobe and Y. Asano. Discharge beginning points of fuel and bleed-air of carburettor 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 carburettor. SAE technical report, 2001.

[7] B. Sendyka and J. Filipczyk, Simulation of the characteristic of a carburettor of an internal combustion engine. SAE technical report, 1995.

[8] D. L. Harrington and J. A. Bolt, Analysis and digital simulation of carburettor metering SAE technical report, 1990.

[9] B. Sendyka and J. Filipczyk, Simulation of the characteristic of a carburettor of an internal combustion engine. SAE technical report, 1995.

[10] B. Sendyka and W. Heydel, The analysis of a constant depression carburettor with a fuel feeder. SAE technical report, 1994.

[11] M. Furuyama and H. Ohgane, A comparison of pulsating and steady flows in terms of carburettor characteristics. JSAE Review, pages 18–23, 1987.

[12] P. J. Moss. Pulsation enrichment in carburettors. Automotive Engineer, pages 53–56, 1980.

[13] B. Wu, Y. Feng, and J. Liu, Numerical simulation of gas-liquid two-phase flow in motorcycle carburettor. 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