International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Special Issue, Vol. 1, No. 1, January 2014
National Conference on "Recent Advances in Mechanical Engineering" RAME – 2014

Research Paper

CFD ANALYSIS OF THE NATURAL GAS BASED CARBURETOR FOR A TWO STROKE SPARK IGNITION ENGINE

Harshwardhan Sharma[1]*, Satyendra Singh[2] and Ravi Goel[1]

In this paper Computational Fluid Dynamics (CFD) analysis has been carried out to study the flow characteristics of methane and air in the existing carburetor designed for a small capacity two stroke S.I engine. Then based upon the results obtained from the above analysis a new prototype carburetor having two gas inlets was designed. Further analysis was then performed on the prototype carburetor to study the effect of using two fuel inlets on the air-fuel ratio. The results of the simulation shows that a two hole carburetor give better fuel distribution characteristics than the existing carburetor

Keywords: CFD, Nozzle, Carburetor

INTRODUCTION

Biogas (mainly methane CH_4) is the best fuel for substituting conventional fuels because of low pollutants and Carbon dioxide emission. Recently sky-rocketing fuel cost, energy security and environmental pollution issues are becoming very important concerns worldwide, so amongst the various alternatives fuels, CNG is the most practical solution. Studies related with the application of natural gas as an alternative fuel in diesel engines has been extensively investigated using CFD analysis, but there is a dearth of research related with the application of natural gas in smaller capacity gasoline engines using CFD analysis. This paper examined the effect of fuel placement technique on the flow behavior of the natural gas in the specially designed prototype carburetor for a two stroke scooter engine. The geometry of the existing carburetor has been slightly modified to meet the air and fuel requirement similar to the gasoline counterpart. The complete CFD analysis was performed on the *ANSYS WORKBENCH V14.0* software. The CFD's predictions showed that the uniformity of fuel distribution was heavily affected by the location of fuel inlets.

[1] Quantum School of Technology Roorkee, Uttarakhand, India.
[2] Kumaon Institute of Technology, Dwarahat, Uttarakhand, India.

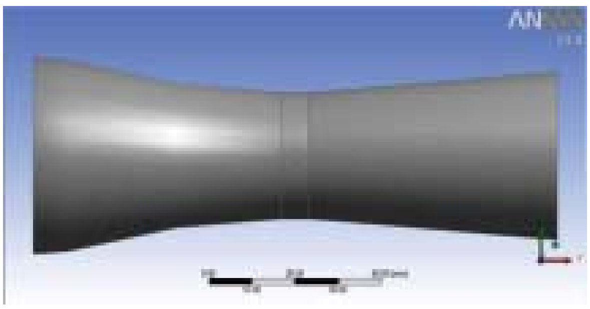
CFD ANALYSIS OF THE EXISTING CARBURETOR

The computational model of the existing carburetor was created in the *Design Modeler* software (as shown in Figure 1) using the measured dimensions from the original carburetor, which are given in Table 1.

Table 1: Dimensions of the Existing Carburetor

Name of the Part	Dimension
Total length of carburetor	120 mm
Inlet diameter	42 mm
Throat diameter	27 mm
Outlet diameter	37 mm
Length of throat	6 mm
Length of inlet section	57 mm
Length of outlet section	57 mm
Fuel nozzle diameter	2 mm

Figure 1: Computational Model of the Existing Carburetor

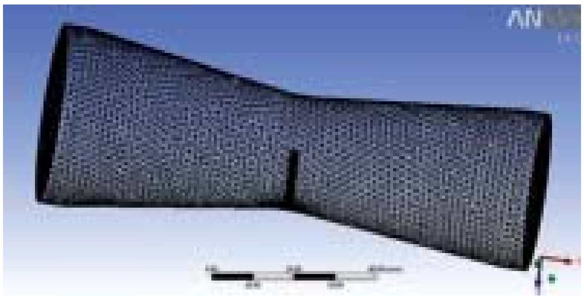


Air enters the geometry at a low speed and for a static pressure specified at the holes, causes an induction of the methane gas due to the low static pressure at the throat due to high dynamic pressure. The mixture of air and methane then flows downstream along the diverging section and encounters an opening type outlet boundary at a fixed static pressure. The challenge in this problem has been the determination of the inlet velocity which is based on engine flow rate calculations. In the present case, a single cylinder 4 stroke 150 cm^3 engine (typical displacement volume for Bajaj Chetek) has been considered with a volume efficiency of 0.85 operating at different engine speeds. The flow rate at the airinlet will be determined as prescribed by standard practices followed in engine calculations (Tatterson, 1994).

Meshing

Computational Fluid Dynamic (CFD)'s modeling predicts essential performance characteristics such as pressure loss, flow rate and temperature distribution [3]. With the help of modeling software, the effect of alternate design ideas on these performance characteristics can be studied to get better efficiency from the final design. CFD analysis includes the numerical solution of the working equations of conservation of mass, momentum, energy, individual species and turbulence model on computational grid or mesh of the selected geometry. In the pre-processing stage inputs of a flow problem to a CFD program by the help of user interface is given and then the consequently the solver converts this input into the suitable form used by it. The prototype carburetor geometry is meshed in *ANSYS ICEM CFD* (dedicated meshing tool) using a tetrahedral mesh as shown in Figure 2.

Figure 2: Tetrahedral 3D Mesh of the Existing Carburetor



Boundary conditions

The followings boundary conditions were used in the Simulations (Geankoplis, 1995):

- Turbulent flow: Turbulence intensity and Hydraulic diameter were given as input. k-å transport equation was used in this study.
- Mixing flow without reaction: In this simulation, it is assumed that there is no reaction occurs between air and natural gas.
- Incompressible: In the incompressible flow, the density of fluids is constant.
- Air Inlet: Fixed inlet velocity condition was considered
- Natural Gas inlet: Fixed static pressure inlet boundary condition was used at the fuel inlets
- Outlet: Fixed Pressure outlet boundary condition was considered.

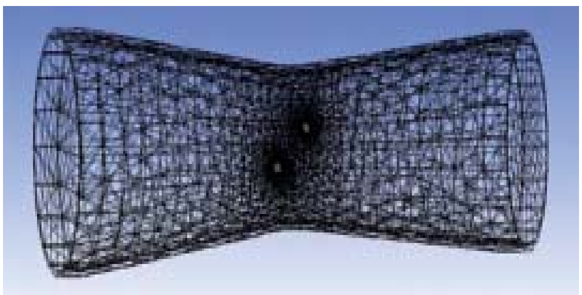
The fully turbulent flows inside the prototype carburetor geometry encouraged the use of standard κ-ε model. In this model the solution of two different transport equation allows the turbulent velocity and length scales to be separately determined. Robustness, economy and reasonable accuracy for a wide range of turbulent flows of the standard ê-å model in *FLUENT* explain its popularity in industrial flow and heat transfer simulation (Geankoplis, 1995). For the convenience of analysis and visualization, surfaces in the domain are needed to display results in a 3D model. Although *FLUENT* creates surfaces for all boundary zones automatically, a self-defined 2D plane at the center of the mixer is created and further analysis is based on it.

CFD Analysis of the Existing Carburetor

The computational model of the prototype carburetor was first drawn in the *Design modeler* software then a 3D tetrahedral mesh of the geometry was created with help of *ICEM.* Figures 3 and 4 shows the computational model and the meshed geometry of the prototype carburetor.

Figure 3: Computational Model of the Prototype Carburetor

Figure 4: 3D Tetrahedral Mesh of the Prototype Carburetor

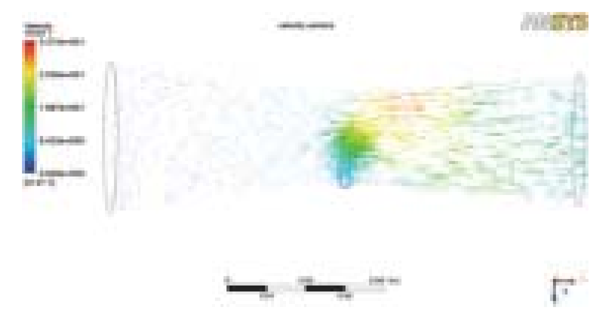


RESULTS AND DISCUSSION

Velocity Vectors for the Model of the Existing Carburetor

Atmospheric air was inducted into the mixer by means of an engine suction pressure of during the suction stroke. The air velocity at inlet section of carburetor was calculated by using procedure explained in (1). The axial velocity prediction in Figure 5 clearly showed that the internally-generated flow aerodynamics of the mixer was governed by mixer design. The flow was expanded upon entering the outlet section of the carburetor, forming re-circulation zones due to obstruction of air flow by fuel injector. These so called “stagnant” flow regions are undesirable characteristics of the ideal carburetor as they may trap fuel and hence may affect the degree of mixture uniformity.

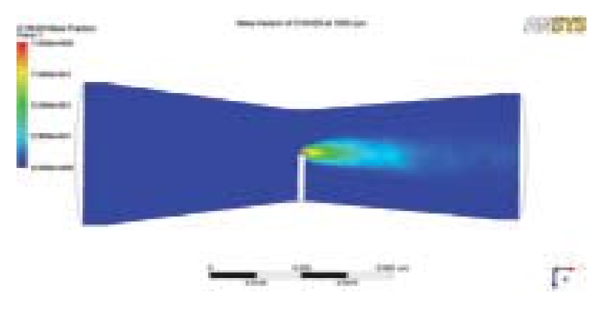
Figure 5: Velocity Vectors for the Existing Carburetor Model



Mass Fractions of the Species for the Existing Carburetor Model

The inlet air was assumed to enter the carburetor at normal temperature and the pressure was taken to be 1 atm. Figure 6 shows the distribution of gasoline at the mid section of the carburetor and figure 6 shows the graphical representation of gasoline distribution along the axis of the carburetor.

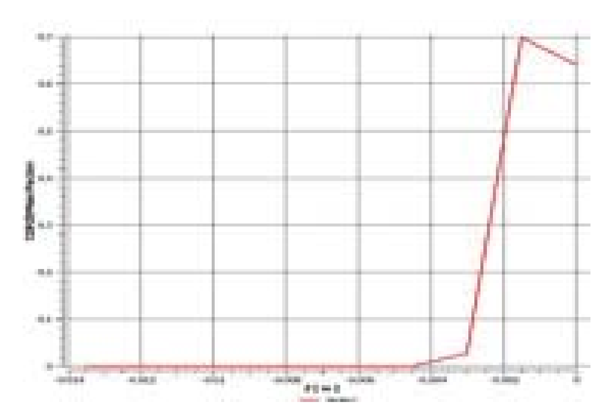
Figure 6: Mass Fraction of Gasoline at the Mid Plane



Variation of Air-Fuel Ratio Across Carburetor Geometry

Figure 7 shows the variation of air-fuel ratio (from bottom end of the carburetor) at different axial locations measured from the inlet of the carburetor. The air-fuel ratio profile just above the injection point at X= 61mm clearly indicates a very rich air-fuel mixture, since the fuel was quickly swept further downstream of the carburetor by high axial air flow. At the upper wall region of the mixer, the CFD results indicate very rich mixtures at all axial locations and virtually very low rate of fuel and air mixing took place in this region. The poor rate of air and fuel mixing resulted in a very

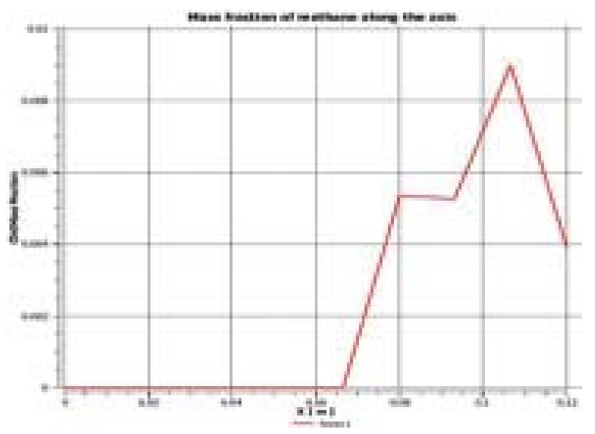
Figure 7: Variation of the Gasoline Mass Fraction Along the Radial Direction at X=61 mm



inhomogeneous mixture charging into the engine. Figures 8 shows the variation of mass fraction of methane along axial direction, which clearly shows the poor distribution of methane. This factor may have contributed to the lower engine performance.

Figure 8 given below shows the variation of mass fraction of methane along the axial direction inside the carburetor geometry.

Figure 8: Variation of Mass Fraction of Methane Along the Axis of the Carburetor



CFD ANALYSIS OF THE PROTOTYPE CARBURETOR

Velocity Vectors for the Prototype Carburetor

The internal flow dynamics in the prototype carburetor was simulated at constant temperature condition of 300 k using air. The axial velocity prediction in Figure 8 shows that the internally-generated flow aerodynamics of the carburetor was governed by carburetor venturi design.

The maximum velocity of 27.08 m/s was obtained at the outlet of the carburetor and the velocity of the mixture at the throat section was came out to be 13.54 m/s compared to 6.7 m/s at the inlet side of the carburetor.

Distribution of Methane Across the Carburetor Geometry

Figure 9 shows the velocity vectors for the prototype carburetor colored with the methane mass fractions and Figure 10 shows the distribution of methane across the carburetor geometry on the 2D plane at the mid-section.

Figure 9: Velocity Vectors for the Prototype Carburetor

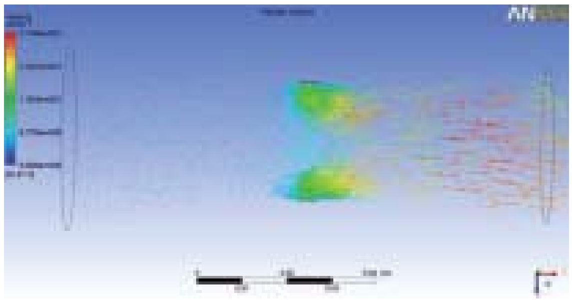
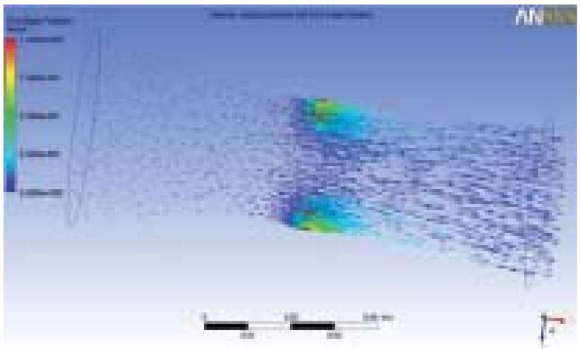


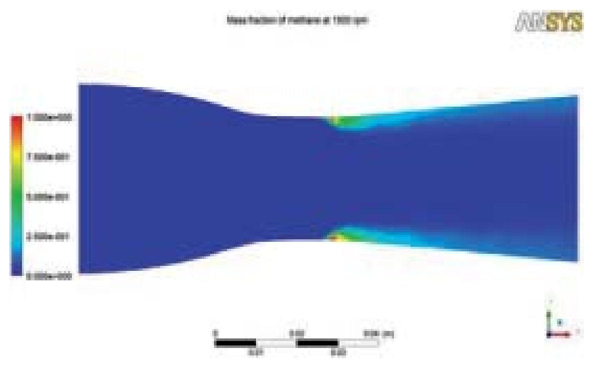
Figure 10: Velocity Vectors Colored with Methane Mass Fraction



As inlet air flows toward the engine, a maximum negative pressure is produced at the throat of the venturi (Anil *et al*., 2006). The gaseous fuel, adjusted and held in a pressure to a large extent equal or near atmospheric pressure (in this case, 34.47 kPa) by the help of a regulator is installed between the gas cylinders and the carburetor fuel inlet.

Figure 10 shows that fuel, which is drawn from the 2 equally spaced nozzles at the throat of the prototype carburetor, is quickly diluted by the stream of air entering through the mixer, as the color contour of mass fraction of methane represents Radial mixing profiles for the prototype carburetor in Figure 11 radial variation (from bottom end of the carburetor) of methane mass fraction at different axial locations downstream of the mixer entrance shows a marked improvement in the distribution of fuel compared to the existing carburetor. At the most upstream where x=61mm, although the mixture is quite lean at the center of the mixer and rich at the wall region, the prevailing turbulence has rapidly mixes the supplied fuel with the charging air. Mixture eventually reaches near optimum ratio at the mixer's exit due to the robust surrounding.

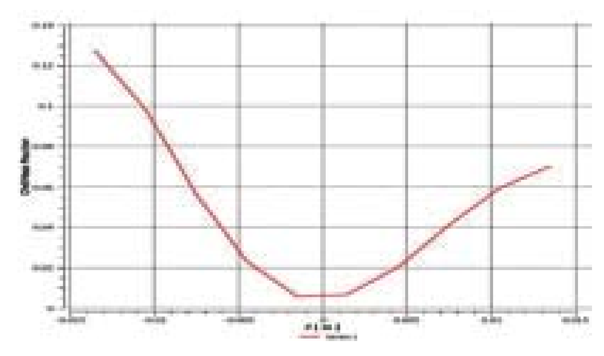
Figure 11: Contours of Methane Mass Fraction Mid Plane



Above graphs shows that the mixture is rich in the regions close to the walls and on the contrary, mixture is quite lean near the axis of the carburetor.

The variation in the mixture strength remains almost constant at the exit of the carburetor, which is a good sign of even distribution or homogeneity of the fuel. Figure 12 given below shows the variation of methane mass fraction at the exit of the prototype carburetor.

Figure 12: Mass Fraction of Methane at the Exit of the Carburetor



CONCLUSION AND RECOMMENDATION

The prototype carburetor is proven to be the best replacement for the existing carburetor when the engine is operating on natural gas for optimum performance, vindicated by results and analysis from computational fluid dynamics software. Now for supply of fuel into the carburetor, a specially designed fuel line, which is encapsulated around the throat's periphery where two gas inlet holes being positioned, is to be fitted in the fuel system jointly with the carburetor. As the methane gas at high pressure flows through the regulator system, this reduces its pressure considerably before being supplied onto the carburetor. A ring shape fuel line around the throat will inject fuel in tandem into the incoming air streams at the throat section of the carburetor. The fuel controlling system will ensure continuous supply of fuel, with the correct ratio into the combustion chamber when the scooter engine is operating on different speeds and loads.

ACKNOWLEDGMENT

I would like to thank my supervisor Professor Satyendra Singh for his support and expertise. I would also like to thank my colleagues Mr. Ravi Goel for their support during the project.

REFERENCES

1. Anil T R, Ravi S D, Shashikanth M, Tewari P G and Rajan Weaver N K S (2006), "CFD Analysis of a Mixture Flow in a Producer Gas Carburetor", *International Conference on Computational Fluid Dynamics, Acoustics, Heat Transfer and Electro Magnetics CFEMATCON*, Andhra University, Visakhapatnam , India.
2. Geankoplis C J (1995), "Transport Processes and Unit Operations", Prentice Hall, Inc., Singapore.
3. Noor M M *et al*. (2000), "Development of a High Pressure Compressed Natural Gas Mixer for a 1.5 liter CNG-Diesel Dual Engine", Universiti Malaysia Pahang, Kuantan, Pahang, Malaysia.
4. Tatterson G B (1994), "Scale up and Design of Industrial Mixing Processes", McGraw-Hill, Inc., USA.
5. Yeap Beng Hi, Zulkefli bin Yaacob, Azeman bin Mustafa (2002), "Computational Investigation of Air-Fuel Mixing System For Natural Gas Powered Motorcycle", *Gas Technology Center (Gasteg)*, Universiti Technologi, Malaysia.
6. Yusaf T Zamri M *et al.* (2000), "Development of a 3d CFD Model to Investigate the Effect of the Mixing Quality on the CNG-Diesel Engine Performance", COE, UNITEN, Kajang, Selangor, Malaysia.

APPENDIX

Nomenclature
SI = Spark Ignition
2D = Two Dimensional
3D = Three Dimensional
κ-ε = K-epsilon turbulent model