CFD Analysis of a Mixture Flow in a Producer Gas Carburetor for optimizing the design configuration

T.R.Anil[1], S.D.Ravi[2], M.Shashikanth[2], P.G.Tewari[3] and N.K.S.Rajan[4]

[1] Research student, and Assistant Professor Dept. of Mechanical Engg. MMEC Belgaum.

[2] M-tech student Visvesvaraya Technological University Belgaum.

[3] Professor Energy Systems, Dept. of Mechanical Engg. BVBCET Hubli.

[4] Principal Research Scientist CGPL, Dept of A.E IISc.

ABSTRACT

Design of a carburetor for producer gas application with special reference for reduced loss of pressure is taken up to generate the optimal fuel–air mixture to meet different load conditions of the engine as well as for varying operating conditions of producer gas reactor. A specially designed producer gas carburetor is comprehensively analyzed for its mixing performance and response with a CFD modeling. The model is made up of a mixer chamber that has the essential orifices for air and fuel (producer gas) inlets to generate stable stoichiometric mixture at near to ambient conditions using the induction of the engine as the driving pressure differential for the flow, and tested for a case of engine of 25 kWe capacity. The CFD simulations are carried out followed with experimental studies to validate the analysis. The results show a consistency in the experimental data and the modeling has provided a good insight into the flow details and has paved way in optimization in the geometrical design to get a good mixing efficiency.

Keywords: computational fluid dynamics, producer gas, carburetor, air/fuel ratio, turbulence

NOMENCLATURE

μ molecular viscosity,

μ_T turbulent viscosity

$C_{\varepsilon 1}$ constant =1.45

$C_{\varepsilon 2}$ constant 1.9.

σ_k turbulent model constant for kinetic energy =1

σ_ε constant for k-ε model =1.3.

INTRODUCTION

In the current state of technological advances, it is recognized that Biomass is one of the viable and sustainable renewable resources and new technologies emerging out of biomass based gasification systems find a significant role in bridging the energy crisis. The advanced biomass gasification systems are known to generate producer gas as the combustible fuel that is clean enough to be used in Direct Injection gas engines. However in order to adapt standard gas engines few of its components need modifications before they are used in the biomass power plants. Since this area is an emerging one and the technology has not been disseminated to the scale of driving market, it is essential that specialized components that require modification need be studied. Carburetor is one of the important components in such Category and it is identified that additional research work is to be carried out in establishing a design procedure for this application. [3]. The work presented here is an effort in this regard.

Air/fuel ratio characteristic exert a large influence on exhaust emission and fuel economy in Internal Combustion engine.[5]. With increasing demand for high fuel efficiency and low emission, the need to supply the engine cylinders with a well defined mixture under all circumstances has become more essential for better engine performance. Carburetors are in general defined as devices where a flow induced pressure drop forces a fuel flow into the air stream [1]. An ideal carburetor would provide a mixture of appropriate air-fuel (A/F) ratio to the engine over its entire range of operation from no load to full load condition. To ensure proper performance, Carburetors should be reproducible and have unequivocal adjustment procedures.

CFD software used for cold flow analysis is CFX 10.The k-ε turbulence model is most commonly used and is considered to be the best model between computational time and precision. [2]. The geometric model is built using Ansys workbench.

PRODUCER GAS CARBURETOR

Mixing devices for gases used in gas engines generally referred to as carburetor, for mixing air and gaseous fuels are commonly attached to the intake manifold of an internal combustion engine. In gas carburetor the mixing of air and gaseous fuels needs to be in a proper ratio for a particular engine load and speed. In designing the producer gas carburetor, simplicity and ruggedness have always been considered as a basic requirement to achieve easy adjustment and reproducible performance. The effective area reduction of gas and air entry holes is considered by taking a suitable coefficient of discharge. The air and fuel flow is through orifices into the mixing chamber of the carburetor which has baffle plates that enables proper mixing of air and fuel. The producer gas carburetor is being designed to have air and fuel flow at ambient conditions to be stoichiometry. The producer gas carburetor is as shown in the Figure 1 has orifices placed at air and gas inlets such that the A/F ratio at ambient flow condition should be stoichiometry for a engine suction pressure of a 25 KWe engine. The amount fuel flow inside the carburetor is controlled by a butterfly valve which is located prior to the fuel inlet orifice. The pressure balancing electronic controller drives suitably the butterfly valve with the help of a motor that brings the valve for a null pressure differential across the manifolds for the fuel and air attached upstream to the main engine manifold and works in suction pressures. If the differential pressure at both the carburetor manifolds are maintained at zero, with the manifolds tuned for their effective flow areas to match the ideal mixture condition, then the mixture flow what we get at engine intake manifold will be stoichiometry. The Figure 1 shown below is the geometric model of the producer gas carburetor designed and analyzed for optimal pressure drop with good mixing ability.
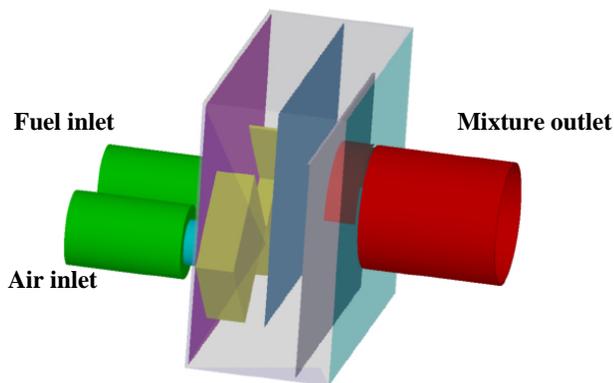

Figure 1: 3D-Model of Producer gas carburetor

3. CFD Analysis

3.1 Problem Definition

 CFD Simulations are carried out on the producer gas carburetor geometric model as shown in Figure 2. The air and producer gas passes through inlet pipe of 50 mm diameter in the axial flow into mixing chamber through an orifice of 28.5mm and 26mm diameter respectively. Producer gas inlet pipe is provided with butterfly valve and geometric model for 0 to 90 in steps of 15 degree is built to carry out analysis at different valve openings. For different mass flow from 0.005kg/s to 0.05 kg/s insteps of 0.005kg/s at the carburetor outlet condition was tried out.

3.2 Grid Generation

The Mesh geometry shown in Figure 3 is a 3-Dimensional model used to simulate the flow analysis in a carburetor with mesh grid density of around 200000 computational nodes is considered. The grid is a unstructured mesh with a maximum mesh grid size of 18mm. Inflated boundary meshes are created for the butterfly valve body and near inlet orifices with first layer mesh size of 1.8mm with an expansion factor of 1.2.
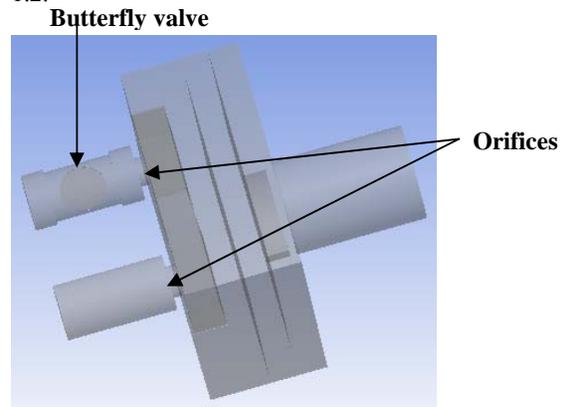

**Figure 2: Geometric model of Producer Gas carburetor**
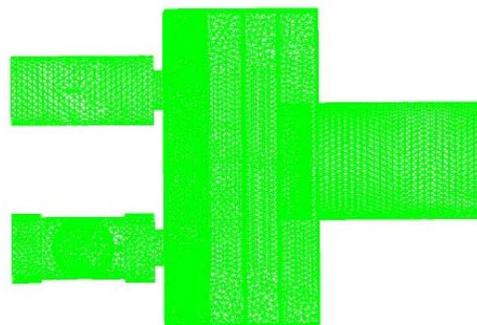
Figure 4: Mesh Geometry Model of Producer Gas Carburetor

3.3 Boundary and Initial Condition

The flow domain considered for simulation is the whole carburetor assembly with steady state flow. Here for simplicity of analysis a single gas (carburetor gas) entity is considered having air ideal gas and producer gas. The relative pressure of the carburetor domain is assumed to be 1 atmosphere with non buoyancy condition. For air inlet boundary condition mass and momentum, static pressure equivalent to domain reference pressure is set with flow condition being subsonic. The initial condition of flow through the air inlet with air ideal mass fraction as 1 is considered. The initial boundary condition for fuel inlet is same as the air inlet except for the flow of producer gas mass fraction being 1 at the inlet. The boundary condition for carburetor outlet is of different mass flow rate which is to be simulated is considered.

3.4 Computational Approach

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional, unsteady and consists of many scales. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number. The k-ε model of turbulence is widely chosen for fluid flow analysis. k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate).

To simulate the turbulence parameters, a standard k-ε model has been chosen with isothermal heat transfer condition at 300 K. The Solver uses k-ε model with two new variables and the continuity equation is then.

$$\frac{\partial \rho}{\partial t} + \nabla \square (\rho U) = 0 \qquad (i)$$

And the momentum equation becomes. [6]

$$\frac{\partial \rho U}{\partial t} + \nabla \square (\rho U \otimes U) - \nabla \square (\mu_{eff} \nabla U)$$

$$= \nabla p' + \nabla \square (\mu_{eff} \nabla U)^T + B \qquad (ii)$$

The flow–solver CFX-10 used for the analysis uses the differential transport equation for the turbulence kinetic energy and turbulence dissipation for analysis.

The equation for kinetic energy K is given by

$$\frac{\partial (\rho \kappa)}{\partial t} + \nabla \square (\rho U k) =$$
$$\nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma k} \right) \nabla_k \right] + P_k - \rho \varepsilon \qquad (iii)$$

The equation for ε without compressibility is given by.

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \nabla \bullet (\rho U \varepsilon) =$$
$$\nabla \bullet \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + \frac{\varepsilon}{K} \left(C_{\varepsilon 1} P_K - C_{\varepsilon 2} \rho \varepsilon \right) \qquad (iv)$$

4. Flow Visualization experimental studies

The model as shown in the plate 2 was constructed by transparent acrylic material and the dimensions of the model are matched to the carburetor that was fabricated and analyzed. The different equipments required to carry out the flow visualization study were developed and fabricated to meet the requirements. The light source with strobed light sheet is positioned over the model so as to have the beam pass through the model in the desired cross section. The particle separation tank is filled with water, and the water with the visual seeding particles (cellulose fiber) is allowed to pass through the carburetor model by using the pump. The flow of water is maintained to get a steady flow. (Typically 20-30 minutes). Then the light source is turned on and strobed at suitable frequency of chopping by a speed controller. The camera is positioned perpendicular to the plane of illumination and the images of the flow visualization are captured. The image captured includes the counter reading of the chopping frequency so that decoding the visual traces is done at the proper time slicing scales.

5. Results

In this section, a contour plot of producer gas mass fraction across a selected plane is presented and also included with two dimensional velocity vectors plots along with actual flow visualization analysis plate are presented.

4.1 Producer gas and air flow mixture analysis

Figure 4 represents the most optimal configuration considered, that shows a three dimensional view of the carburetor, along with a contour plot of the mass fractions within the

mixture over a cross–sectional plane, for same flow conditions as in the earlier cases. The contours of the mass fraction of the producer gas have its color scale with red for 1 (100% fuel) and blue for 0 (100% air). On can see that as the mixture passes across the orifices, baffle plates and the semi–cylindrical plates before it gets out of the carburetor, the mixture is found to have producer gas mass around 0.5 uniformly, representing most desirable condition of mixing for the carburetion.

The Figure 5 shows the flow pattern at the outlet of the carburetor with plot showing the contour for producer gas mass fraction, for a total mass flow rate of 0.025Kg/s and with the condition of the butterfly valve of gas inlet being opened 90º (full). The flow pattern shows the initial mixing pattern of air and producer gas. The design of the mixing paths in the carburetor is optimized with this help and the present configuration is seen to achieve the producer gas is mixed fairly well making its mass fraction to be between 0.2 and 0.3 that is known to be mixtures good enough for premixed combustion.

The figure 6 represents an improved configuration that has enhanced the mixing. The producer gas mass fraction lies around 0.5 of the mixture before it leaves the carburetor. The figure 6 represents the flow pattern when the solver has converged with the residuals of all the variables have diminished to well within the acceptable limits (typically around 10^{-5}).

The engine simulation experimental studies carried out to get a exhaustive analysis of the carburetor performance for varying load conditions . A characteristic curve with air to fuel ratios obtained as an outcome of the study is shown in Figure 7.

The CFD study was extended to get a comprehensive analysis of the performance of the carburetor with varying mass flow rates ranging from 0.005 to 0.05 Kg/s and with different positioning of the control valve between 0 to 90º.

A characteristic curve with air to fuel ratios obtained as an outcome of the study is shown in Figure 8. The chart shows mixture flow at different fuel inlet valve openings and different mass flow rates at outlet condition derived from CFD analysis.

Figure 9 represents the producer gas mass fraction on a line created on the carburetor domain for mixture flow rate of 25g/s and for 90

degree butterfly valve opening. The producer gas mass fraction is around 0.4 & 0.5

The flow field captured by the CFD analysis, taking account of turbulence, in the 3–D domain of the carburetor is shown in the figure 10. The velocities on a plane just before the outlet of the carburetor is around traced to be about 5 m/s and at the centre of outlet on the plane to be around 11 m/s. Two vortices are traced to have formed just upstream of the outlet, due to the restriction by the semi–cylindrical plate introduced between the second baffle plate and the outlet. This arrangement has derived enhanced mixing of the gases and in meeting the desired criteria.

The plate 1 shows the flow pattern in a carburetor that is a photographed picture of a flow visualization technique at frequency of light chopping 2.5H_Z. The flow visualization technique is one of the steps in devising experimental verifications in the analysis during the designing the new carburetor with differential pressure sensing, that enables studying of the flow patterns and to get an account of recirculation inside the carburetor. The flow visualization study was carried out with table top test model made out of acrylic sheets. The experimental results compared with CFD results indicate that the flow patterns captured by the CFD analysis are close to the patterns captured from the experimental results as shown in figure 11, implying that verification for the CFD analysis is complete with consistency among the two[5]
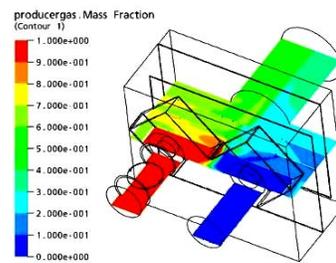

Figure 4: Three Dimensional Contour Plot of on a plane for an outlet mass flow rate of 0.25Kg/s
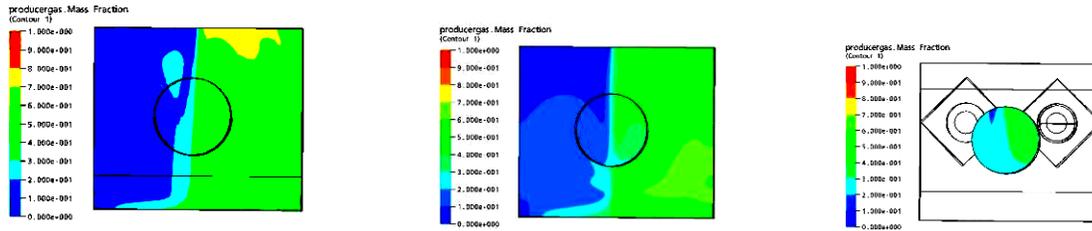
Figure 5: Producer gas mass fraction on plane a) between baffles, b) between baffle plate and outlet c) Outlet for 25 iterations for 90degree Butterfly valve opening and outlet condition 0.025Kg/s
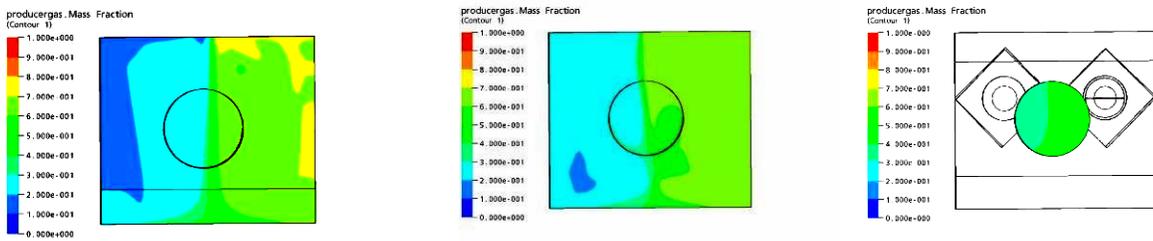

Figure 6: Producer gas mass fraction on plane a) Between inlet and first baffle plate b) Between baffles plate c) Between second baffle plate and outlet d) Outlet for 145 (convergence) iterations for 90degree Butterfly valve opening and outlet condition 0.025Kg/s
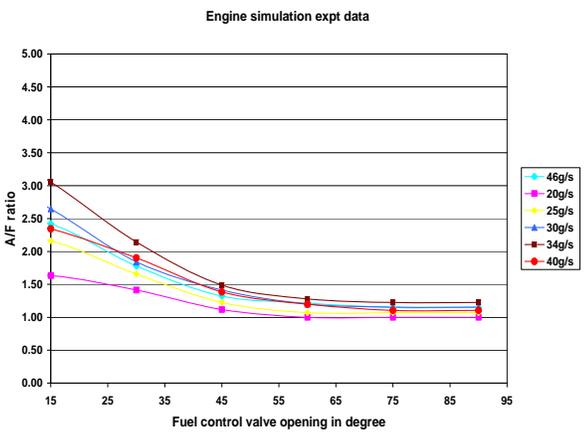

Figure 7 Engine simulation Experimental Results
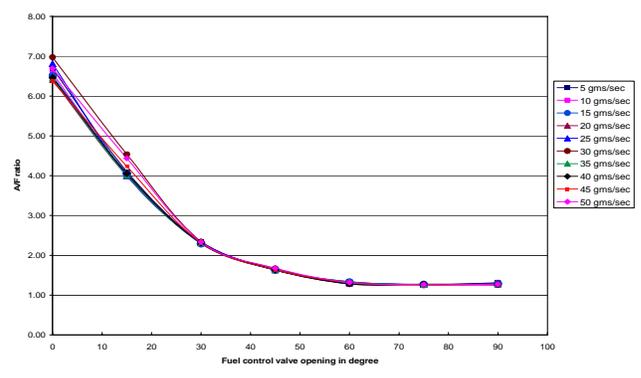
Figure 8 Air Fuel Characteristic curve for different fuel inlet valve opening derived from CFD Analysis
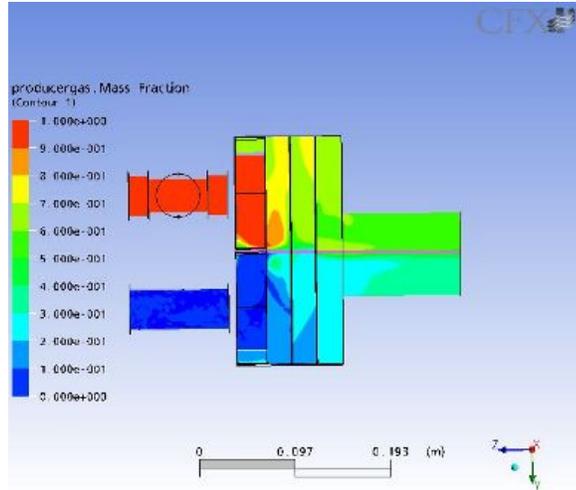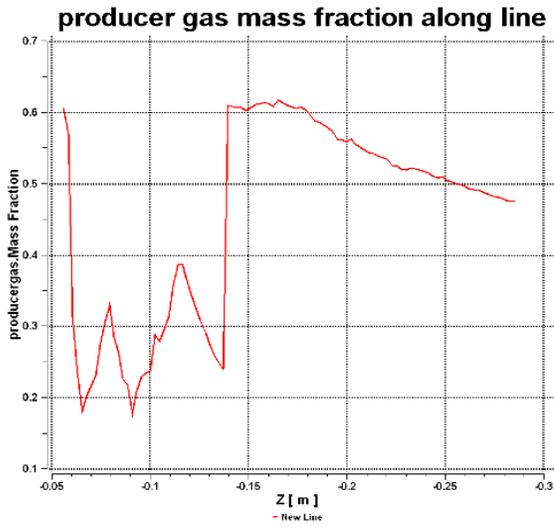
Figure 9 Producer gas Mass fraction on a line for mixture flow rate of 25g/s and for 90 degree butterfly valve opening.
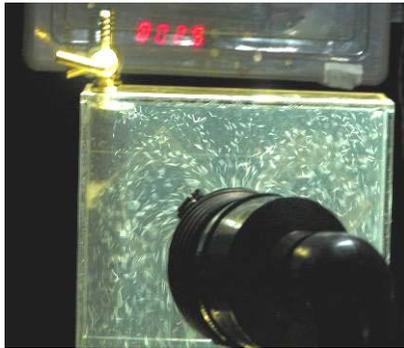

Plate 1
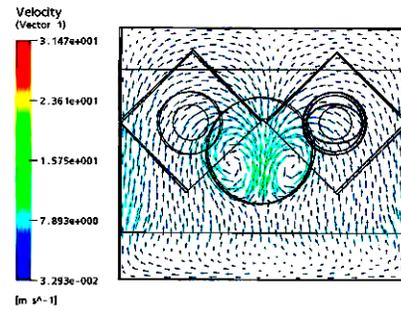
Figure 10
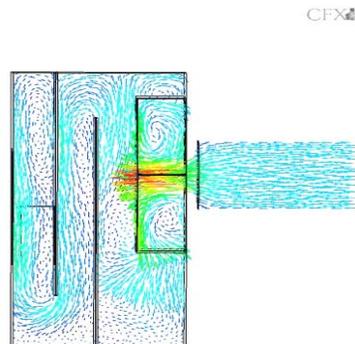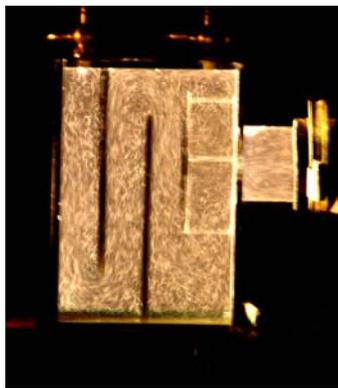
Figure 11: Comparison of flow between experimental and computational results

Differential carburetor model

Air relief taps

Outlet

Inlets
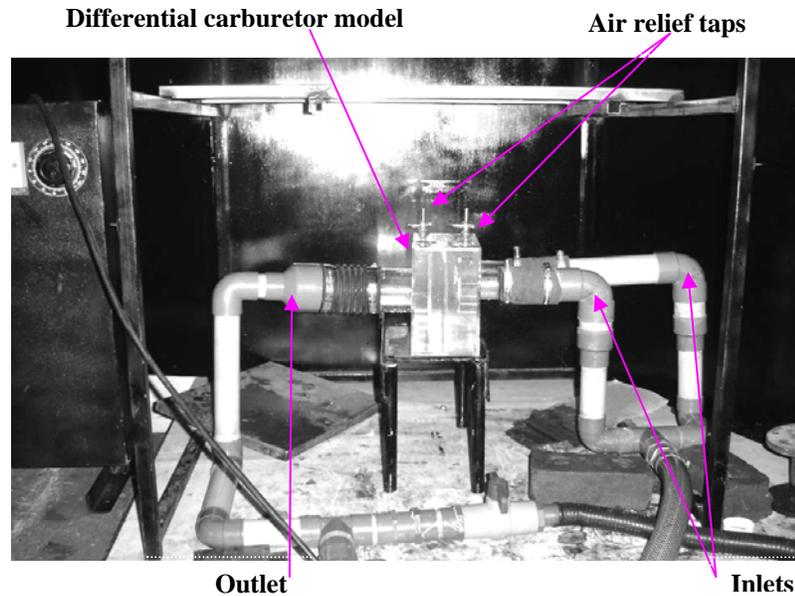
Plate 2 Flow visualization study inside the carburetor model

5. Conclusion

Three dimensional CFD computations on producer gas carburetor made have been able to capture the detailed functional features of fluid flow in the carburetor configurations considered and are found to be consistent with the experimental work done with simulations for engine operating conditions. Turbulent model based on k-ε theory with a RANS code has been used for the CFD predictions of the producer gas mass fraction and the carburetor performance has been evaluated leading to bringing out of an optimal design of the PG carburetor that is used for prototype testing and real–time testing. The experimental results compared with CFD results indicate that the flow patterns captured by the CFD analysis are close to the patterns captured from the experimental results, implying that verification for the CFD analysis is comprehensive enough having consistency among the two studies.

6. References

1   **Klimstra J** "Carburetors for Gaseous Fuels –on Air to Fuel ratio, Homogeneity and Flow restriction. SAE paper 892141

2   **Versteeg, H.K.** and **Malalasekara, w.** An introduction to CFD-The finite volume method, 1995 (Longman Scientific and Technical)

3   **T.R.Anil, P.G.Tewari N.K.S.Rajan**   An Approach for Designing of Producer Gas Carburetor for Application in Biomass based Power Generation Plants proceedings of the national conference of Natcon 2004 Bangalore.

4   **Yoshishige Ohyama**, "Air /fuel ratio control using upstream models in the intake system." SAE paper 0857 vol1 1999.

5   **T.R.Anil, S.D.Ravi, M.Shashikanth, N.K.S.Rajan P.G.Tewari. "***CFD Analysis of a Mixture Flow in a Producer Gas Carburetor"* International Conference On Computational Fluid Dynamics, Acoustics, Heat Transfer and Electromagnetics CFEMATCON-06, July 24-25, 2006, Andhra University, Visakhapatnam – 530003, INDIA

6   **Stephen B Pope, Turbulent Flows, 2000** Cambridge University Press