World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:4, No:11, 2010

Open Science Index, Mechanical and Mechatronics Engineering Vol:4, No:11, 2010 publications.waset.org/1578.pdf

Computational Investigation of Air-Gas Venturi Mixer for Powered Bi-Fuel Diesel Engine

Mofid Gorjibandpy, Mehdi Kazemi Sangsereki

Abstract**—**In a bi-fuel diesel engine, the carburetor plays a vital role in switching from fuel gas to petrol mode operation and vice-versa. The carburetor is the most important part of the fuel system of a diesel engine. All diesel engines carry variable venturi mixer carburetors. The basic operation of the carburetor mainly depends on the restriction barrel called the venturi. When air flows through the venturi, its speed increases and its pressure decreases. The main challenge focuses on designing a mixing device which mixes the supplied gas is the incoming air at an optimum ratio. In order to surmount the identified problems, the way fuel gas and air flow in the mixer have to be analyzed. In this case, the Computational Fluid Dynamics or CFD approach is applied in design of the prototype mixer. The present work is aimed at further understanding of the air and fuel flow structure by performing CFD studies using a software code. In this study for mixing air and gas in the condition that has been mentioned in continuance, some mixers have been designed. Then using of computational fluid dynamics, the optimum mixer has been selected. The results indicated that mixer with 12 holes can produce a homogenous mixture than those of 8-holes and 6-holes mixer. Also the result showed that if inlet convergency was smoother than outlet divergency, the mixture get more homogenous, the reason of that is in increasing turbulence in outlet divergency.

Keywords—Computational Fluid Dynamics, Venturi mixer, Air-fuel ratio, Turbulence.

I. INTRODUCTION

IN the current state of technological advances, it is recognized that biomass is one of the 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 [1]. 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 to 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.

Air-fuel ratio characteristic exert a large influence on exhaust emission and fuel economy in internal combustion engine. 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. An ideal carburetor would provide a mixture of appropriate air-fuel ratio to the engine over its entire range of operation from no load to full load condition [2]. To ensure proper performance, carburetors should be reproducible and have unequivocal adjustment procedures. Turbulence model used in this study is k-ε model and can get the best condition between computational time and precision, for subsonic internal flow [3-5].

II. MIXER DESIGN

The purpose of a mixer is to mix a proper amount of fuel with air before admission to engine cylinder. The mixing is very important in dual-fuel engine, as it will provide a combustible mixture of fuel and air in the required quantity and quality for efficient operation of the engine under all conditions.

In this study the inlet throat and orifice diameters for mixers with 6, 8 and 12 holes were obtained by using of isentropic compressible flow analysis [6]. The length of each mixer was selected arbitrary. Finally the optimized mixer was selected using computational fluid dynamics. The values which are used to study isentropic compressible flow are as below.

Engine efficiency: %34
Engine load: 710KW
Venturi inlet Mach number: 0.05
Air and gas pressure: 10^5Kpa
Air temperature: 300k
Gas temperature: 280k
Holes discharge coefficient: 0.55
Air fuel ratio: 31.5
In Table 1, 2 the mixers geometrical dimensions can be seen.

M. Gorjibandpy. Department of Mechanical Engineering, Babol Noshirvani University of Technology, Babol, Iran (corresponding author to provide phone: 00981113224048; fax: 00981112296582; e-mail: gorji@nit.ac.ir).

M. S. Kazemi. Department of Mechanical Engineering, Babol Noshirvani University of Technology, Babol, Iran (e-mail: mehdi_kazemi123@yahoo.com).

TABLE I
MIXERS GEOMETRICAL DIMENSIONS

Number of orifice	Mixer names	Inlet diameter (cm)	Throat diameter (cm)	Hole diameter (mm)
12	A_{12}	30.4	13.4	11
	B_{12}	30.4	13.4	11
	C_{12}	30.4	13.4	11
8	A_8	30.4	13.4	13
	B_8	30.4	13.4	13
	C_8	30.4	13.4	13
6	A_6	30.4	13.4	15
	B_6	30.4	13.4	15
	C_6	30.4	13.4	15

TABLE II
MIXERS GEOMETRICAL DIMENSIONS

Mixer names	Inlet length (mm)	Convergence Length (mm)	Throat length (mm)	Divergence length (mm)	Outlet length (mm)
A_{12}	600	600	100	300	600
B_{12}	600	450	100	450	600
C_{12}	600	300	100	600	600
A_8	600	600	100	300	600
B_8	600	450	100	450	600
C_8	600	300	100	600	600
A_6	600	600	100	300	600
B_6	600	450	100	450	600
C_6	600	300	100	600	600

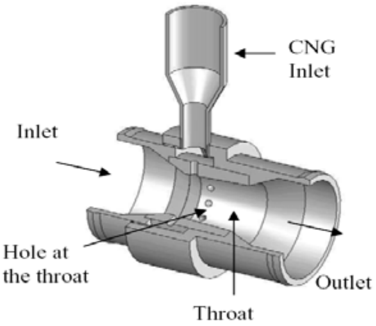

Fig. 1 Venturi Mixer

Figure 1 shows a mixer visage, whereas linkage between gas and air lines is very clear, also we can see the place of holes in throat very well.

III. GOVERNING EQUATION

Turbulence consists of fluctuations in the flow field in time and space and 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 [7]. The k-ε model of turbulence is widely chosen for fluid flow analysis where k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity and 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 this model with two new variables and the continuity equation is as follow [8]:

$$\frac{\partial}{\partial x_j}\left(\rho \overline{u_j}\right) = 0 \tag{1}$$

$$\overline{u_j}\frac{\partial}{\partial x_j}\left(\rho \overline{u_i}\right) = B - \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) - \rho\overline{u_i' u_j'}\right) \tag{2}$$

$$\overline{u_j}\frac{\partial}{\partial x_j}\left(\rho C_p \overline{T}\right) = H + \frac{\partial}{\partial x_j}\left(\rho C_p \alpha \frac{\partial \overline{T}}{\partial x_j} - \rho C_p \overline{u_j' T'}\right) \tag{3}$$

$$\overline{u_j}\frac{\partial}{\partial x_j}\left(\rho \overline{C}\right) = R + \frac{\partial}{\partial x_j}\left(\rho D \frac{\partial \overline{C}}{\partial x_j} - \rho \overline{u_j' C'}\right) \tag{4}$$

$$\rho\frac{\partial k}{\partial t} + \rho u_j k_{,j} = \left(\mu + \frac{\mu_t}{\sigma_k} k_{,j}\right)_{,j} + G + B - \rho\varepsilon \tag{5}$$

$$\rho\frac{\partial \varepsilon}{\partial t} + \rho u_j \varepsilon_{,j} = \left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \varepsilon_{,j}\right)_{,j} + C_1\frac{\varepsilon}{k}G + C_1(1 - C_3)\frac{\varepsilon}{k}B - C_2\rho\frac{\varepsilon^2}{k} \tag{6}$$

IV. THE MODELING SOFTWARE AND BOUNDARY CONDITION

The Computational Fluid Dynamics (CFD) is based on the numerical solutions of the fundamental governing equations of fluid dynamics namely the continuity, momentum, energy, species and turbulent equations [7-9]. The Fluent 6.3.26 software package was used to accomplish this job. This software flow solver is a finite volume, pressure based, fully implicit code solving the 3D Navier-Stokes equations governing fluid flow and associated physics. The code is used for the modeling of a wide range of industrial problems involving fluid flow, heat transfer (including radiation), turbulence, mixing of chemical species, multi-step chemistry, two-phase flows, moving-rotating bodies and other complex physics.

Turbulent flow: This involves the use of a turbulence model, which generally requires the solution of additional transport equations and k-ε transport equation is used in this study. Three quantities; turbulent kinetic energy, K, dissipation rate, D and length scale, L are very important in specify the turbulence characteristics at the inlet. If K and D are specified, the value for L will be ignored. It is sometimes more convenient to provide a lengths scale instead of a value for the dissipation rate. The length scale that would be used for an internal flow is usually the inlet diameter or height.

Mixing flow without reaction: This requires the solution of additional equations for mixture fractions or species mass fraction. In this simulation, it is assumed that there is no reaction between air and natural gas.

Air Inlet boundary: The condition needs to specify in inlet boundaries is the fixed velocity condition. For this condition, the flow solver determines the mass flow rate applied to each face of the boundary using the velocity specification, the pressure, the temperature and the fluid property density.

Fuel inlet boundary: Fixed static pressure inlet boundary condition was used at the fuel inlets. By using this boundary condition type, the mass of the fuel inducted into the venturi will be part of the solution. Fuel will be inducted into the venturi because of the low pressure created at the throat.

Outlet boundaries: The Fixed Pressure outlet boundary conditions serves to anchor the system pressure and allow both inflow and outflow to satisfy continuity in the domain. Since fixed pressure outlet boundaries can also allow inflow, it is important to provide realistic values of turbulence quantities, temperature and mixture at these boundaries even though they are not required. These values are only used to evaluate diffusion at the boundary.

V. RESULTS AND DISCUSSION

In Table 3 air fuel ratio of each mixer can be seen. Since the mixer A_{12} supplies the best air fuel ratio, therefore this mixer is selected as optimized mixer.

TABLE III
THE AIR-FUEL RATIO OF THE MIXERS

Mixer names	A_{12}	B_{12}	C12	A_8	B_8	C_8	A_6	B_6	C_6
Air fuel ratio	31.1	30.3	29.2	34.1	33.1	31.7	34.0	33.6	33.2

The mixing efficiency is very important to determine the quality of combustion. A high combustion quality will produce low exhaust gas emission components. In dual fuel engine, the mixer should be design to be able to produce lean air-fuel mixture, as the diesel engine is compression ignition type running on the lean side [10].

The air supplied with excessive CNG might produce incomplete combustion because the amount of hydrocarbon is high when it mixes with diesel. An incomplete combustion will produce high amount of CO, which is hazardous to the environment. Moreover, the excessive air will lead to high NOx emission.

The mixing quality for 6, 8 and 12 holes venturies has been investigated using CFD. The simulation was done for the same air inlet velocity and same outlet pressure of the real engine geometry.

Comparing the three venturies, it can be seen that the mixing is more homogeneous for the 12-hole mixer.

Methane mass fraction in all mixers is shown in figures 2, 3 and 4. The figures show that inlet convergency is smoother than outlet divergency because of increasing turbulence.

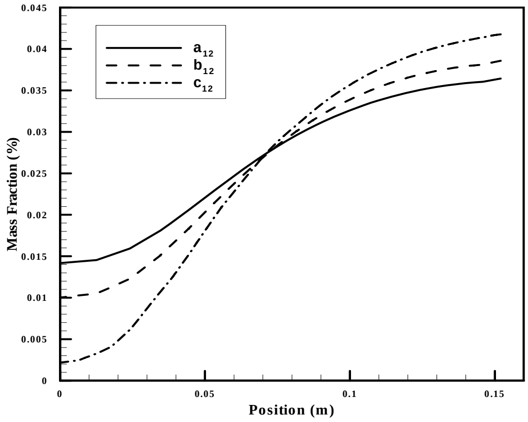

Fig. 2 Methane mass fraction in the outlet of 12-holes mixer

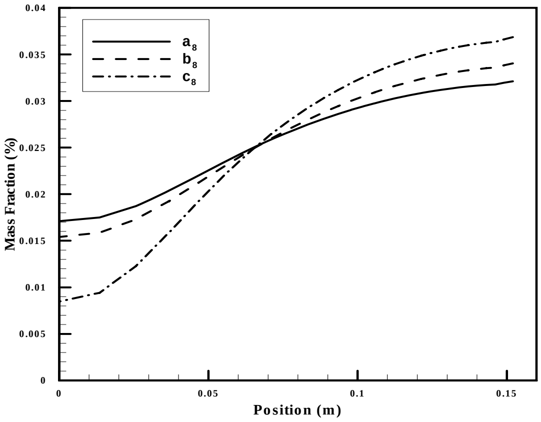

Fig. 3 Methane mass fraction in the outlet of 8-holes mixer

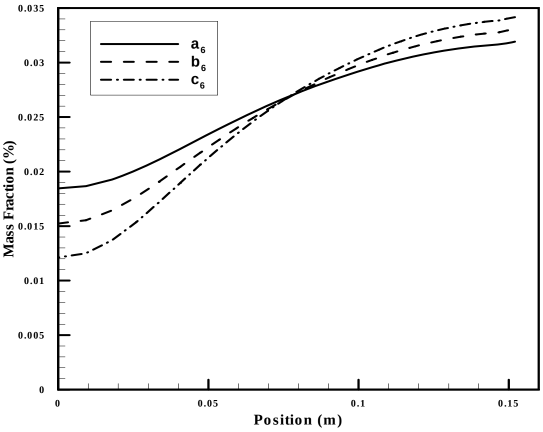

Fig. 4 Methane mass fraction in the outlet of 6-holes mixer

The contour of pressure, velocity, temperature, turbulence, mass fraction, in 12-holes mixer is shown figures 5-11. As we can see in figure 10, the effective area of current is less than geometrical area.

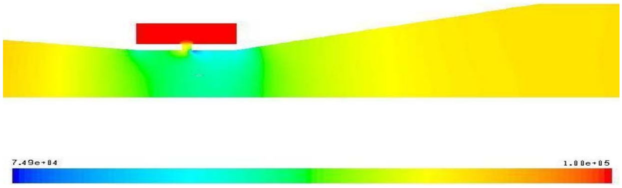

Fig. 5 Contour of pressure in 12-holes mixer (pa)

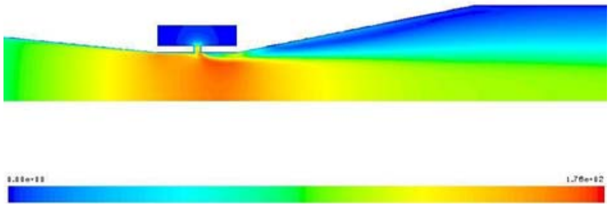

Fig. 6 Contour of velocity magnetude in 12-holes mixer (m/s)

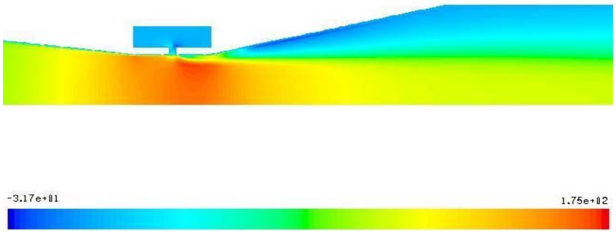

Fig. 7 Contour Contour of velocity in flow direction in 12-holes mixer (m/s)

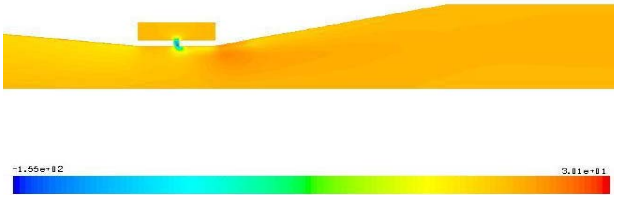

Fig. 8 Contour of velocity in perpendicular to flow direction in 12-holes mixer (m/s)

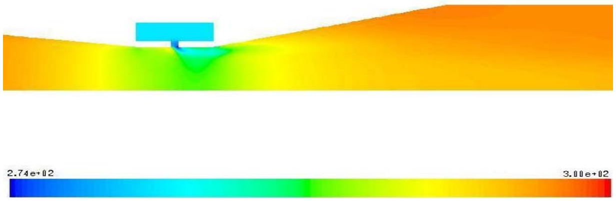

Fig. 9 Contour of temperature in 12-holes mixer (k)

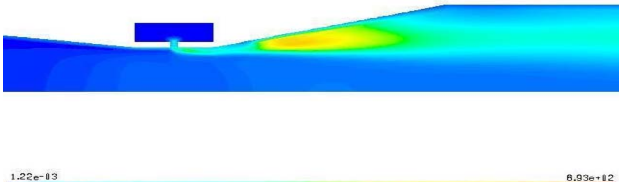

Fig. 10 Contour of turbulent kinetic energy in 12-holes mixer (m^2/s^2)

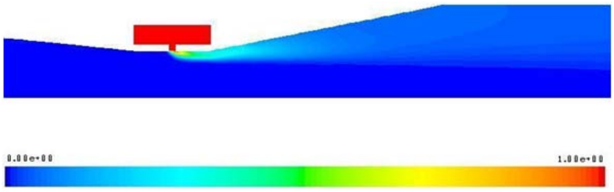

Fig. 11 Contour of mass fraction of mathane in 12-holes mixer (%)

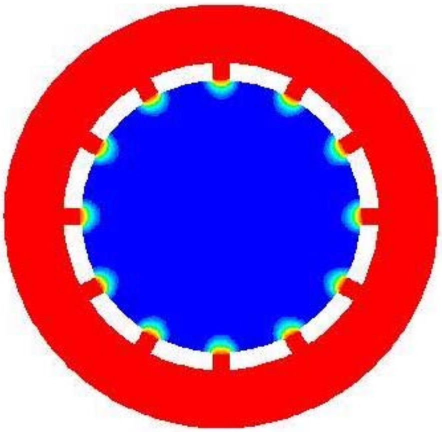

Fig. 12 Contour of mass fraction of mathane in 12-holes mixer (%)

VI. CONCLUSION

A CNG-air venturi mixer for CNG-diesel dual-fuel stationary engine has been designed using 3D CFD analysis. Venturies with four and eight holes have been simulated. It was found that 12-hole venturi mixer gives better mixing performance as compared to 8-hole and 6-hole venturi mixer. The emission of CO2 was less for dual-fuel compared to diesel at all operating conditions. A high quality mixer should provide the very optimum environment for the engine operating condition. In the process of designing the CNG mixer, it can be deducted that the design has reached the objectives. Air flow is one of the areas that need to be considered seriously when a CNG mixer is designed, as it not only reduce the pollution by automobile vehicle but also improves the vehicle performance. The engine emissions level is also very important as we designed and created the engine by using methane as a fuel and this will improve the emission as we discussed earlier. A high quality mixer should provide the very optimum environment for the engine operating condition. In the process of designing the CNG mixer, it can be deducted that the design has reached the objectives.

REFERENCES

[1] Maxwell T. T. and Jones J. C., "Alternative Fuels: Emissions, Economics and Performance", USA Society of Automotive Engineers, SAE Inc, 1995.

[2] Devarajan R., "Comparative Evaluation of a Two Stroke Compressed Natural Gas Mixer Design using Simulation and Experimental Techniques", International Conference on Association of Science and Technology for Development, Power and Energy Systems, Langkawi, 359-362, 2008.

[3] Yusaf T. F., "Development of a 3d CFD Model to Investigate the Effect of the Mixing Quality on the CNG-Diesel Engine Performance", 1995.

[4] Smith K., Arellano L. and Jerry I. "Tustaniwskyj Designing a Fuel/Air Mixer to Reduce Emissions", University of California, San Diego Department of Mechanical Engineering, USA, 2006.

[5] Mardani H., "Development of Intake System for Improvement of Performance of Compressed Natural Gas Spark Ignition Engine", Universiti Teknologi Malaysia, 42-47, 2004.

[6] Liepmann W. H. and Roshko a., "Element of Gas Dynamics", John Wiley and Sons, Inc, 1960.

[7] Raghunathan B. D. and Kenny R. G., "CFD simulation and validation of the flow within a motored two stroke engine", International Congress and Exposition, Detroit, Michigan 1997, SAE 970359, 1997.

[8] Xu B. Y. and Furuyama M., "Visualization of Natural Gas-Air Mixing Flow in the Mixer of a CNG Vehicle", Technical Notes, JSAE Review, 18 57 82, 1997.

[9] Rahim A. R., "Design and Simulate Mixing of Compressed Natural Gas with Air in a mixing device", Malaysian Technical Universities Conference on Engineering and Technology, Perlis, Vol. 2, 99-104, 2008.

[10] Ganesan. V, "Internal Combustion Engine", Second edition, McGraw-Hill Book Company, 2004.