CFD OPTIMIZATION OF VENTURI MIXER DESIGN IN BI-FUEL GAS ENGINES

MR. PRASANNA SUTAR
PG Scholar, M.Tech I.C. Engines, Vel Tech University Chennai - ARAI Academy, Pune.
E-mail: prass.sutar@gmail.com

Abstract-The carburettor is the most important part of the fuel system of a gas engine. In a bi-fuel gas engine, the carburettor plays a vital role in switching from fuel gas to petrol mode operation and vice versa. All gas engines carry variable venturi mixercarburettors. The basic operation of the carburettor mainly depends on the restricting 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 in the incoming air at an optimum ratio. In order to overcome 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 prototypemixer. The present work is aimed at a 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 16 holes can produce a homogenous mixture than those of 12-holes and 10-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- Air-fuel ratio, Computational Fluid Dynamics, Venturi mixer, 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. Carburettor 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. Carburettors are in general defined as devices where a flow induced pressure drop forces a fuel flow into the air stream. An ideal carburettor 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, carburettors 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 10, 12 and 16 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.

In Table 1 & 2 the geometrical dimensions of mixers can be seen.

| TABLE I | MIXERS GEOMETRICAL DIMENSIONS |
|---|---|---|---|---|
| Number of Orifice | Mixer Names | Inlet Diameter (mm) | Throat Diameter (mm) | Hole Diameter (mm) |
| 16 | A16 | 50 | 22.5 | 2.0 |
|   | B16 | 50 | 22.5 | 2.0 |
|   | C16 | 50 | 22.5 | 2.0 |
| 12 | A12 | 50 | 22.5 | 1.5 |
|   | B12 | 50 | 22.5 | 1.5 |
|   | C12 | 50 | 22.5 | 1.5 |
| 10 | A10 | 50 | 22.5 | 2.5 |
|   | B10 | 50 | 22.5 | 2.5 |
|   | C10 | 50 | 22.5 | 2.5 |
TABLE II
MIXERS GEOMETRICAL DIMENSIONS

<table>
<thead>
<tr>
<th>Mixer Names</th>
<th>Inlet Length (mm)</th>
<th>Convexence Length (mm)</th>
<th>Throat Length (mm)</th>
<th>Divergence Length (mm)</th>
<th>Outlet Length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>A15</td>
<td>13</td>
<td>20</td>
<td>10</td>
<td>18</td>
<td>13</td>
</tr>
<tr>
<td>B15</td>
<td>13</td>
<td>15</td>
<td>10</td>
<td>22</td>
<td>13</td>
</tr>
<tr>
<td>C15</td>
<td>13</td>
<td>12</td>
<td>10</td>
<td>25</td>
<td>13</td>
</tr>
<tr>
<td>A25</td>
<td>13</td>
<td>20</td>
<td>10</td>
<td>18</td>
<td>13</td>
</tr>
<tr>
<td>B25</td>
<td>13</td>
<td>15</td>
<td>10</td>
<td>22</td>
<td>13</td>
</tr>
<tr>
<td>C25</td>
<td>13</td>
<td>12</td>
<td>10</td>
<td>25</td>
<td>13</td>
</tr>
<tr>
<td>A30</td>
<td>13</td>
<td>20</td>
<td>10</td>
<td>18</td>
<td>13</td>
</tr>
<tr>
<td>B30</td>
<td>13</td>
<td>15</td>
<td>10</td>
<td>22</td>
<td>13</td>
</tr>
<tr>
<td>C30</td>
<td>13</td>
<td>12</td>
<td>10</td>
<td>25</td>
<td>13</td>
</tr>
</tbody>
</table>

Fig. 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.

The values which are used to study isentropic compressible flow are as below.

- Engine efficiency: 34 %
- Engine load: 7.65 kW
- Venturi inlet Mach number: 0.05
- Air and gas pressure: $10^7$ Kpa
- Air temperature: 300 K
- Gas temperature: 280 K
- Holes discharge coefficient: 0.55
- Air fuel ratio: 20.6

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 inertial 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 ε 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 300K. The Solver uses this model with two main variables and the continuity equation is as follows [8]:

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \quad (1)$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = B - \frac{\partial}{\partial x_j}(\frac{\mu}{\sigma_k} \frac{\partial u_i}{\partial x_j} - \rho u_i u_j) \quad (2)$$

$$\frac{\partial}{\partial x_j}(\rho C_p \epsilon) = \frac{\partial}{\partial x_j}(\rho D \frac{\partial \epsilon}{\partial x_j}) - \rho u_i u_j \frac{\partial \epsilon}{\partial x_j} + \frac{\partial}{\partial x_j}(\rho C_p \epsilon) \quad (3)$$

$$\rho \frac{\partial \epsilon}{\partial x_j} + \rho u_i k_j = \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial u_i}{\partial x_j} + G + B - \rho \epsilon \quad (4)$$

$$\rho \frac{\partial \epsilon}{\partial x_j} + \rho u_i e_j = \left( \mu + \frac{\mu_t}{\sigma_k} e_j \right) + C_1 \frac{\epsilon}{k} G + C_2 (1 - C_3) \frac{\epsilon}{k} B - C_4 \epsilon \frac{\epsilon}{k} \quad (5)$$

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 14.5 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 modelling of a wide range of industrial problems involving fluid flow, heat transfer (including radiation), and 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 to 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 A16 supplies the best air fuel ratio, therefore this mixer is selected as optimized mixer.

<table>
<thead>
<tr>
<th>Mixer names</th>
<th>A16</th>
<th>B15</th>
<th>C14</th>
<th>A12</th>
<th>B12</th>
<th>C12</th>
<th>A20</th>
<th>B10</th>
<th>C13</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air fuel ratio</td>
<td>19. 1</td>
<td>19. 9</td>
<td>21. 2</td>
<td>21. 8</td>
<td>22. 6</td>
<td>21. 1</td>
<td>20. 9</td>
<td>21. 6</td>
<td>1</td>
</tr>
</tbody>
</table>

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 designed to be able to produce lean air-fuel mixture.

The air supplied with excessive CNG might produce incomplete combustion because the amount of hydrocarbon is high when it mixes with gasoline. 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 10, 12 and 16 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 16-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.

The contour of pressure, velocity, temperature, turbulence mass fraction, in 16-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. 2 Methane mass fraction in the outlet of 16-holes mixer](image)

![Fig. 3 Methane mass fraction in the outlet of 12-holes mixer](image)

![Fig. 4 Methane mass fraction in the outlet of 10-holes mixer](image)

![Fig. 5 Contour of pressure in 16-holes mixer (pa)](image)
CONCLUSION

A CNG-air venturi mixer for dual-fuel engine has been designed using 3D CFD analysis. Venturis 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 gasoline 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 deduced 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 levels also very important as we designed and created the engine using methane as a fuel and this will improve the emissions as 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 deduced that the design has reached the objectives.

REFERENCES


★★★