Modeling of diesel/CNG mixing in a pre-injection chamber

H A Abdul-Wahhab, A R A Aziz, H H Al-Kayiem, and M S Nasif

Abstract. Diesel engines performance can be improved by adding combustible gases to the liquid diesel. In this paper, the propagation of a two phase flow liquid-gas fuel mixture into a pre-mixer is investigated numerically by computational fluid dynamics simulation. CNG was injected into the diesel within a cylindrical conduit operates as pre-mixer. Four injection models of Diesel-CNG were simulated using ANSYS-FLUENT commercial software. Two CNG jet diameters were used of 1 and 2 mm and the diesel pipe diameter was 9 mm. Two configurations were considered for the gas injection. In the first the gas was injected from one side while for the second two side entries were used. The CNG to Diesel pressure ratio was varied between 1.5 and 3. The CNG to Diesel mass flow ratios were varied between 0.7 and 0.9. The results demonstrate that using double-sided injection increased the homogeneity of the mixture due to the swirl and acceleration of the mixture. Mass fraction, in both cases, was found to increase as the mixture flows towards the exit. As a result, this enhanced mixing is likely to lead to improvement in the combustion performance.

1. Introduction

Due to the large occurrence in a variety of natural and practical applications, multiphase turbulent mixtures have attracted significant engineering interest. These applications include liquid-gas fueled combustors, liquid fuel injected combustion engines, mixtures propellant combustion and fire suppression and control. All of these situations are involving dispersed phase species which considerably affects the thermochemical nature of the surrounding carrier phase. Therefore, mixing and evaporation processes in liquid fuel combustors have marked effects on ignition characteristics [1]. 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. Air-fuel ratio characteristics exert a large influence on exhaust emissions and fuel economy in internal combustion engine. With increasing demand for high fuel efficiency and low emissions, the need to supply the engine cylinders with a well-defined mixture under all circumstances has become more essential for better engine performance [2]. A liquid-gas fuels mixer is one of the important components in such application and it is identified that additional research work need to be carried out in establishing a design procedure for this application.

To ensure proper performance, the liquid-gas mixer should be reproducible and has unequivocal adjustment procedures. For that, a Diesel-CNG mixer has been simulated and this paper presents the simulation results at various design configurations and operational conditions. The simulation was carried out using that ANSYS-FLUENT version 14 software. The turbulence model used in this study is k-ε model which represents a compromise between computational time and precision, for subsonic internal flow [3-5].
2. Mixer design and computational modelling

The purpose of the pre-mixer is to achieve a proper homogenous two phase fluid of liquid Diesel with gas CNG before injection into the engine cylinder. The two phase mixing is very important in dual-fuel engine, as it will provide a combustible mixture of liquid-gas in terms of the required quantity and quality for efficient operation of the engine under all operation conditions. In this study, six designs of mixers labeled as SA1, SA2, DB1, DB2, SC1, and DD1 have been modeled and simulated. The design variables were different CNG jet diameters (1 mm and 2 mm) and the injection angle of CNG into the core of the mixer (55° and 90°). The 90° is selected as it provides the best tangential flow of the jet in the mixture. Then, we selected the 55° as it represents almost the midway between the perfect tangential and the normal (0°). We haven’t select the 45° where the flow will create jet flow in the mixture with high turbulence without enhancement to the flow to swirl in the mixer body. The length of the mixer was selected arbitrarily as 60 mm. Table 1 shows the geometrical dimensions of each design.

| Mixer label | Diesel pipe (dp) diameter (mm) | CNG jet diameter (dinj) (mm) | Length L (mm) | Injection angle β_inj |
|-------------|-------------------------------|-----------------------------|---------------|-----------------------|
| SA1         | 9                             | 1 (single)                  | 60            | 90°                   |
| SA2         | 9                             | 2 (single)                  | 60            | 90°                   |
| DB1         | 9                             | 1 (double)                  | 60            | 90°                   |
| DB2         | 9                             | 2 (double)                  | 60            | 90°                   |
| SC1         | 9                             | 1 (single)                  | 60            | 55°                   |
| DD1         | 9                             | 1 (double)                  | 60            | 55°                   |

Figure 1 shows a model of single injection mixer (SA1), where the diesel and CNG inlets are identified clearly. The liquid diesel is injected axially while the gaseous CNG is injected tangentially.

![Figure 1](image)

3. Governing equation

Turbulence consists of fluctuations in the swirl 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 the viscous forces, and is characterized by a high Reynolds number [6] and [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 and swirl analysis, a standard k-ε model was chosen with isothermal heat transfer condition
at 300 K. The Solver uses this model with two new variables and the continuity equation as follows [8]:

the continuity equation:

$$\frac{\partial}{\partial x_j} (\rho \overrightarrow{u_j}) = 0$$  \hspace{1cm} (1)

the momentum equation:

$$\overrightarrow{u_j} \frac{\partial}{\partial x_j} (\rho \overrightarrow{u_i}) = \overrightarrow{B} - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \frac{\partial \overrightarrow{u_j}}{\partial x_i} + \frac{\partial \overrightarrow{u_i}}{\partial x_j} \right) - \rho \overrightarrow{u_i} \overrightarrow{u_j}$$  \hspace{1cm} (2)

the energy equation:

$$\overrightarrow{u_j} \frac{\partial}{\partial x_j} (\rho c_p T) = H + \frac{\partial}{\partial x_j} \left( \rho c_p \alpha \frac{\partial T}{\partial x_j} - \rho c_p \overrightarrow{u_j} \cdot \nabla T \right)$$  \hspace{1cm} (3)

the model with two new variables:

$$\overrightarrow{u_j} \frac{\partial}{\partial x_j} (\rho \overrightarrow{c}) = R + \frac{\partial}{\partial x_j} \left( \rho D \frac{\partial \overrightarrow{c}}{\partial x_j} - \rho \overrightarrow{u_j} \cdot \nabla \overrightarrow{c} \right)$$  \hspace{1cm} (4)

$$\rho \frac{\partial k}{\partial t} + \rho u_j k_{1,j} = \left( \mu + \frac{\eta_k}{\sigma_k} \right) \frac{\partial}{\partial x_j} \left( k_{1,j} \right) + G + B - \rho \varepsilon$$  \hspace{1cm} (5)

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho u_j \varepsilon_{1,j} = \left( \mu + \frac{\eta_k}{\sigma_k} \right) \frac{\partial}{\partial x_j} \left( \varepsilon_{1,j} \right) + c_1 \frac{\varepsilon}{k} G + c_2 \left( 1 - c_3 \right) \frac{\varepsilon}{k} B - c_2 \rho \varepsilon^2$$  \hspace{1cm} (6)

where, j=1, 2, 3, which are refereeing to three axis.

4. Computational modelling

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, two phase flow [9]. The FLUENT 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 considering incompressible fluids. 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. Figure 2 shows the injection angle and the mesh for cases under study.

The three-dimensional model of the mixer is developed by using the preprocessor CFD software. The number of cells used in this model (SA2) was 316,177. Mesh refinement investigation has been carried out to optimize the number of cells used. It has been found that increasing the number of cells to 377,697 and 411,045 has no effect on the results accuracy. Hence, 316,177 were selected to be the optimum number of cells that can be used in the simulation.
4.1. Turbulent flow
This involves the use of a turbulence model, which generally requires the solution of additional transport equations and k-\(\varepsilon\) transport equation is used in this study. Three quantities; turbulent kinetic energy, k, dissipation rate, D and length scale, L are very important in specifying the turbulence characteristics at the inlet in two phase flow cases. 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 pipe length.

4.2. Mixing flow without reaction (Two Phase Flow)
This requires the solution of additional equations to two phase flow for mixture fractions or species mass fraction. In this simulation, it is assumed that there is no reaction between Diesel and natural gas. Table 2 Properties of Diesel Fuel and CNG.

| Parameter | Diesel Fuel | CNG |
|-----------|-------------|-----|
| Density (kg/m\(^3\)) \(20\,^\circ\text{C}\) | 830 | 0.72 |
| Viscosity (Cst) \(20\,^\circ\text{C}\) | 6 | \(7.8\times10^{-6}\) |
| LCV (Mj/kg) | 43.8 | 45.8 |
| Octane Number | 50 | 125 |
| Cloud point \(^\circ\text{C}\) | -9 | -27 |
| Flash point \(^\circ\text{C}\) | 70 | 220 |

4.3. Diesel Inlet boundary
The condition needed to be specified at inlet is the pressure. For this condition, the flow solver determines the pressure applied to each face of the boundary using the velocity specification, the mass flow rate, the temperature and the fluid density. Table 3 initial conditions for the simulation.

4.4. CNG inlet boundary
Static pressure inlet boundary condition was used at the CNG inlet with ratio of 1.5 to 3 from Diesel pressure inlet. This higher gas pressure than the liquid pressure is the practical case to prevent back flow in the gas injector. This range of Gas-to-liquid pressure is reasonable to simulate the mixing phenomena. The mass mixing ratio of CNG to Diesel is selected as 0.7 and 0.9. This mixing ration is
the common practice in the steady of the gas to liquid mixing for fuel application, as reported by [3, 5, 10].

| Parameter                  | Initial Condition |
|----------------------------|-------------------|
| Diesel inlet pressure (bar)| 2                 |
| CNG inlet pressure (bar)   | 3 to 6            |
| Injection temperature (K)  | 300               |
| Wall temperature (K)       | 300               |
| Mass CNG/Diesel ratio      | 0.7 to 0.9        |

5. Result and discussion
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 engines, the mixer should be designed to be able to produce lean air-fuel mixture in the cylinder, as the diesel engine is a compression ignition which is usually produce lean air/fuel mixture [11]. In this work, the idea of pre-mixing CNG-Diesel was used to increase the air quantity inside the cylinder more than if a mixture of CNG-Air is injected in a diesel fuelled engine which may result is a rich mixture. An incomplete combustion will produce high amounts of CO, which is hazardous to the environment. Moreover, the excessive air will lead to high NOx emission. The mixing quality for the 6 injection models was investigated using CFD. The simulation was done for the same diesel inlet pressure and same outlet pressure of the real engine geometry. Comparing the six mixers, it can be seen that the mixing is more homogeneous for the double injection mixer with jet diameter 1 mm and injection angle 55°. CNG mass fraction in all mixers is shown in figures 3 and 4. Figure 3 shows the results due to the change of the injection angle from 90° to 55° and decreasing the jet diameter from 2 mm to 1 mm. The results demonstrate that the mixing is more homogeneous for the single, 1 mm mixer compared to single 2 mm mixer. While for the double injection case, shown in figure 4, shows that inlet with double injection side is smoother than single injection side because of the increased swirl and turbulence.

![Figure 3](image_url)

Figure 3. CNG mass fraction (X) in single injection mixers SA₂, SA₁ and SC₁.
The contours in Figure 5 show the variation of mixing pressure in cases of one inlet and two inlets at 90° and 55° injection angle for each case. If the CNG flow rates increase at certain points there is a tendency of the flow to attain an annular type flow regime, which is beneficial to obtain dispersed bubble and droplet populations. It is found that when CNG injection, in both cases of injection angles, that the injection at 55° impose the swirl to be born far from the wall. In contrast, the 90° injection enhances the swirl and initiates the swirl to initiate near the wall. This will lead to proper dispersion of the CNG in the Diesel fuel.

Figures 6 and 7 show the contours of the velocities in the mixer for 2 mm and 1 mm CNG injector diameters, respectively. The simulation results for 2 mm single injection side, shown in figure 6, is misleading where the values and intervals of streamlines are not the same. Rotation in the plane seems stronger and more extended for the 2 mm double injection side, since much shorter intervals of stream
function values have been employed, in order to be better detected. Thus, what could be seen with the green and blue areas of figure 6 at SA2 are only weak vortexes that consequently encompass large areas. Figure 7 The flow predicted by the simulation in the case of the mixer with double injection at 1 mm diameter and injection angle 55°, actually rotates much more vigorously. It is clearly exhibiting vortexes with concentrated rings and of a stronger inner structure. Finally, it should be noted that the prediction of mixer DD1 is almost axisymmetric, which is in agreement with the boundary conditions, barring the surely minor detail of the primary swirl. In contrast, the results of mixer SC1 showed an asymmetric flow, especially near the throat, in a manner that cannot be related easily to the geometry.

On the other hand, it can be noticed from the simulation results in these figures that the bubbles coalescence and break-up are continuous processes in the transport phenomena of the two-phase flows in the mixer. If the liquid flow rate increases, the associated liquid turbulent dissipation is noticed to increase. This increase is attributed to the break up mechanism of the bubbles due to severe turbulent splitting. As a result of the predominant splitting action, the relatively large bubbles in the flow will divide into smaller almost spherical bubbles. With a decrease in the volume of the bubble, the interfacial tension dominates maintaining the spherical bubble shape. Under such extreme conditions, dispersed bubbles or homogeneous flow patterns would exist, even at high void fractions. Thus, highly turbulent dissipation and breaking is the key mechanism responsible for flow pattern transition in turbulent two-phase flows. From the contours of relative velocity angle shown in figures 6 and 7, it can be noticed that this phenomenon increases when mixing by using double side with 1mm diameter nozzle and injection angle 55° because of the increase in turbulence due to the increased swirl.

Figure 6. Contours of the velocity (m/s) in injector diameter 2mm to: (SA2) single injection side, (DB2) double injection side.
Figure 7. Contours of the velocity (m/s) in injector diameter 1mm to: (SC) single injection side, (DD) double injection side.

Figure 8 shows the validation of the simulation procedure, where a comparison was made with the experimental results of Rankin, et al. [10]. They carried out the experiments of co-axial CNG injector at mass flow ratio 0.7 with double injection mixer (DD1) at 1 mm injector diameter with mass flow ratio of 0.7 and 0.9. The figure compares our simulation results with their experimental results in terms of the axial velocity of the mixture.

Figure 8. Comparing the results with Rankin, et al. [10].
6. Conclusions
A liquid-gas mixer for Diesel-CNG dual-fuel stationary engine was studied using 3D CFD analysis. Mixers with single and double injections and different jet diameters and injection angles were simulated. It was found that the double-sided gas injection mixer gave better mixing performance as compared to the single injection mixer. Also, the smaller jet diameter produced better mixing due to the increased jet velocity and enhanced turbulence. A high quality mixer should provide the very optimum environment for the engine operating condition.

Acknowledgments
The authors acknowledge Universiti Teknologi PETRONAS for providing the facilities existing in center for automotive research and energy management (CAREM). The main author expresses his thankful remarks to the Iraqi government for the financial support to pursue his PhD.

References
[1] Boivin M, Simonin O, Squires K D 1998 J Fluid Mech 375 235–63
[2] Devarajan R 2008 International Conference on Association of Science and Technology for Development Power and Energy Systems (Langkawi) (Langkawi Power and Energy Systems) p. 359-62
[3] Yusaf T, Yusoff M Z 1995 Proceedings of the International Conference and Exhibition and Natural Gas Vehicles
[4] Smith K, Arellano L, Jerry I, Tustaniwsky J 2006 University of California, San Diego Department of Mechanical Engineering, USA)
[5] Mardani H 2004 Universiti Teknologi Malaysia) 42-7
[6] Sutar P 2003 IRF-International Conf. Organized by Institute of Research (Deemed University, Pune)
[7] Raghunathan B D, Kenny R G 1997 International Congress and Exposition (Detroit, Michigan )
[8] Xu B Y, Furuyama M 1997 Technical Notes, JSAE Review 18 57-82
[9] Rahim A R 2008 Malaysian Technical Universities Conference on Engineering and Technology (Perlis) vol 2 99-104
[10] Rankin B A, Guilden Becher D R, Gore J P 2012 Spring Technical Meeting of the Central States Section of the Combustion Institute 22–4
[11] Ganesan V. Internal Combustion Engine. Internal Combustion Engine Second edition: McGraw-Hill Book Company; 2004.