Prakash et al. 2017. *Int. J. Vehicle Structures & Systems*, 9(5), 284-287
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.9.5.04

International Journal of
Vehicle Structures & Systems
Available online at www.maftree.org/eja

Flow Behaviour Analysis of Injection Mixer for CNG Engines using Computational Fluid Dynamics

R. Paullinga Prakash[b], S. Palani[b], D. Vijaya Kumar[b], S. Arun Kumar[a] and S. Shanmugam[c]

[a]*Dept. of Mech. Engg., Apollo Engg. College, Chennai, Tamilnadu, India*
Corresponding Author, Email: arunsangeeth1022@gmail.com
[b]*Dept. of Mech. Engg., Vel Tech Multitech, Avadi, Tamilnadu, India*
[c]*Dept. of Physics, Vel Tech Multitech, Avadi, Tamilnadu, India*

ABSTRACT:

Use of gaseous fuels for fuelling the engines reduces reactive hydrocarbons and do not pose the problem of vaporization as with the liquid fuels. One of the problems of gaseous mixers is the ability to prepare a homogeneous mixing of air and fuel at a specific air-fuel ratio prior to entering the engine resulting high exhaust emissions. The objective of this project is to carry out three dimensional CFD analysis of CNG injection mixer to understand the flow behaviour of air fuel mixture and to optimize the design of injection mixer. The analysis was carried out by varying the injection position and injection inclination. The results of the CFD simulation could be used to understand the effect of position of fuel tube, injection inclination in the mixing of air and fuel. Further the results of the study would also be considered for the design modification.

KEYWORDS:

Flow analysis; Computational fluid dynamics; Injection mixer; CNG engines

CITATION:

R.P. Prakash, S. Palani, D.V. Kumar, S.A. Kumar and S. Shanmugam. 2017. Flow Behaviour Analysis of Injection Mixer for CNG Engines using Computational Fluid Dynamics, *Int. J. Vehicle Structures & Systems*, 9(5), 284-287. doi:10.4273/ijvss.9.5.04

1. Introduction

One of the problems of gaseous mixers is the ability to prepare a homogeneous mixing of air and fuel at a specific air-fuel ratio prior to entering the engine. This issue, if not being taking care of, may result in high BSFC and high exhaust emissions. Hence, investigation was conducted to enhance the mixing in Throttle Body Injection Mixer (TBIM), which is the new generation of mixer for a CNG motorcycle, through CFD simulation. The purpose of this study is to prepare a homogeneous mixing of air and fuel at a specific air-fuel ratio before entering the engine. Air-fuel mixing in a direct injection spark ignition (DISI) engine [1], measurement of in-cylinder mixing rate [2], simulation of the interaction of intake flow and flow spray in a DISI engine [3], simulation and control of CNG engines [4], optimisation of air-fuel mixing homogeneity and performance improvements of a stratified-charge direct injection combustion system [5] and so forth were reviewed. In this research, CFD simulation was conducted to determine and optimise the injection frequency in TBIM for the best air-fuel mixing prior to entering the engine.

As yet, the research works available in improving the mixing quality of injection-type mixers include the study of geometry design, injection to cross flow velocity ratio, cross flow swirl strength, injection timing, wall-impinging injection with a bump placed in the injection impingement region, injection position and injection inclination angles [6]. Study on the effect of various injection frequencies on mixing is still rarely seen. Hence, to ensure the simulation condition was compatible with the actual engine operating condition, experimental work was conducted at the outset of the research to obtain the engine suction pressure in the intake manifold for each case study. The data was then verified thoroughly through previous work before applying it in the CFD simulation [6]. The same research methodology had been carried out to investigate the effect of various injection inclination angles on the mixing in TBIM and its results were found to be consistent with previous work [6].

Concern over the consistency of CFD simulation results with the corresponding theoretical and experimental findings has been an issue since the last few years. Nevertheless, the performance of CFD simulation has shown a dramatic improvement after consecutive refinements over the years. Ref [3] claimed that CFD simulation was an equal partner with pure theory and pure experiment in the analysis and solution of fluid dynamic problems. Other researches such as [1], [5] and [7] have also proven the validity of the results obtained through CFD simulation. In addition, CFD simulation of the single-phase mixing problems has been well-established [3]. The objective of this project is to carry out three dimensional CFD analysis of CNG injection mixer to understand the flow behaviour of air fuel mixture and to optimize the design of injection mixer. Thus, the results obtained in this simulation work can be utilised with a very high level of confidence.

2. CFD simulation of throttle body

Air-fuel mixer is a device where fuel is metered and mixed with the incoming air in accordance with engine requirements. Owing to the dissatisfaction in the mixing homogeneity and air-fuel ratio control of the existing air-fuel mixer, a new air-fuel mixer called Throttle Body Injection Mixer (TBIM), which is of electronic fuel controlled, was developed. A schematic diagram of TBIM is shown in Fig. 1. It consists of a throttle valve, fuel injector, reducer and intake manifold. The amount of air entering TBIM is manipulated by the opening of throttle valve whereas the amount of fuel required is controlled by the fuel injector. The air and fuel are mixed in TBIM before entering the engine. In this work, the fuel is injected by placing the injector at various positions along the length of the mixer and also the injector is positioned with vertical, 15° and 30° inclinations. CFD simulation of TBIM commenced with model meshing, followed by and selection of turbulent model before starting the computation.

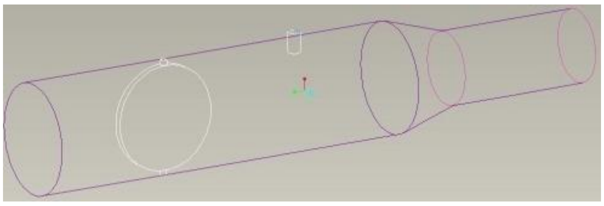


Fig. 1: Wire fame model of injection mixer

2.1. Meshing

Mesh generation is the process by which spatial discretisation of CFD model is accomplished. Meshing is based on tetrahedron element discretisation. There are two types of grids to choose from for model meshing, i.e. structured and unstructured grids. The first one is composed of hexahedral elements while the latter one consists of tetrahedral elements. A general rule of thumb is to apply the structured grids on simple geometries and the unstructured grids on complex geometries. For the main body of TBIM, which consists of a throttle valve and an injector, the unstructured grids were used as shown in Fig. 2. On the other hand, the structured grids were chosen for simple geometries like the intake manifold (an arched cylinder) and the reducer (a pinnacle-less cone) that connects the main body of TBIM with the intake manifold Fig. 2.

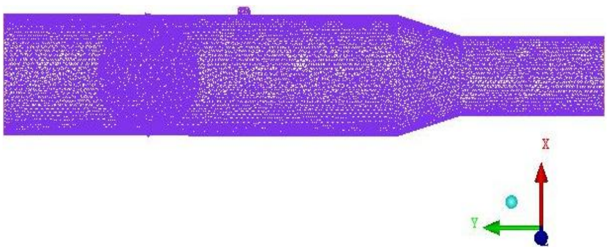


Fig. 2: Meshed model

A general guideline of model meshing is to always mesh the more critical domains (i.e. high velocity, high pressure or pressure drop flow fields) in advance of the less critical ones. For TBIM, the more critical domains include the regions around the openings of throttle valve and the injector Fig. 2. The fineness of grid applied increases with the criticalness of a domain. After TBIM was appropriately meshed, the boundary conditions of TBIM were identified for the air inlet, fuel outlet, mixture outlet and the symmetry plane. Boundary conditions of 'pressure inlet' and 'mass flow inlet' were selected for the air inlet and fuel outlet, whereas the mixture outlet and the symmetry plane were given the boundary condition of 'wall'. These choices of boundary condition were depending on the available data and also the data required from the simulation.

2.2. Turbulence model

Among all the turbulent models, the k-ε model was selected in this work because it is the most widely used and validated one in terms of consistency and reliability [8-9]. There are three types of k-ε model, namely the Standard k-ε model, Renormalization Group (RNG) k-ε model and Realizable k-ε model. The major differences between these models are the methods in calculating the turbulent viscosity, turbulent Prandtl numbers and the generation and destruction terms in the ε equation. The Standard k-ε model is the most extensively used and validated turbulent model among all the k-ε models. It has achieved notable successes in calculating a wide variety of thin shear layer flows. The transport equations of k-ε used in this analysis are,

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

where ρ is the fluid density, k is the kinetic energy, ε is the dissipation rate and σ_k, σ_ε, $C_{l\varepsilon}$ and $C_{2\varepsilon}$ are constants.

3. Stoichiometric air fuel ratio

The term stoichiometric ratio describes the chemically correct air-fuel ratio necessary to achieve complete combustion of the fuel. In this analysis, air and fuel were delivered into TBIM at stoichiometric air-fuel ratio, i.e. 17.3:1. Nevertheless, the air-fuel ratio of the mixing before entering the engine could be differed from this value owing to poor mixing. A good mixing would give a stoichiometric mass fraction of methane ($M_{CH4.S}$) locally throughout the mixing region. The $M_{CH4.S}$ was calculated as $M_{CH4.S}$ = 1/18.3 = 0.0546.

4. Results and discussion

4.1. Mass faction of CH_4

Mass fraction of CH4 refers to concentration of CH_4 in the mixture of air and CH_4. Stoichiometric mass fraction of methane has been calculated as 0.055. Fig. 3 shows the distribution of mass fraction of CH_4 in the mixing region and outlet of injection mixer for the various injection inclinations. It has been observed from the results that for vertical injection position mass fraction of methane decreases in the near wall towards outlet. 15° injection position shows increased distribution of mass fraction near wall region as well as at outlet. The results of 30° injection position shows that comparatively increased mass fraction of methane in the wall regions as well as towards outlet than other injection positions.

4.2. Inter phase mass transfer rate

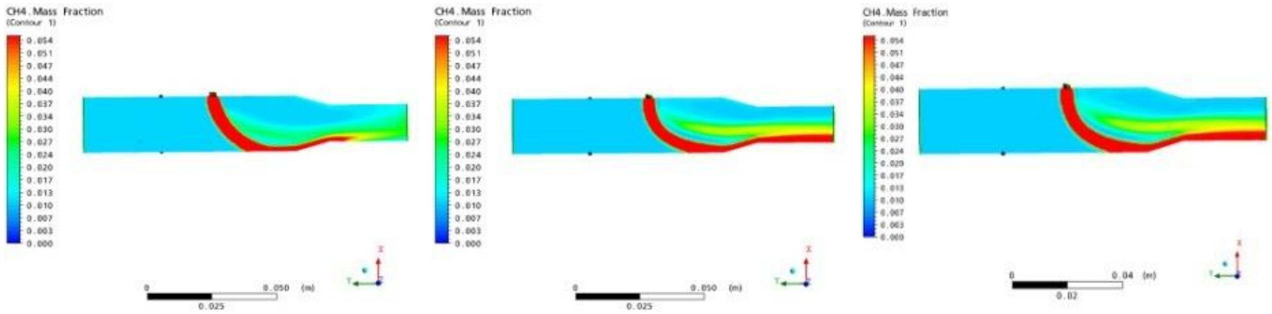
Inter phase mass transfer rate refers to rate of mass transfer between the air and CH_4 after which is expressed as kg/s.m^3. Better the mass transfer rate would result in good mixing along the flow. Fig. 4 shows the distribution of mass transfer rate after the injection occurs. For the vertical injection position mass transfer rate decreases along the flow from the mixing region towards the outlet. The decreased rate is observed in the near wall region after nozzle. A slight increase in the mass transfer rate observed in 15° injection position. The 30° injection position shows comparatively increased rate in the near wall region as well as at outlet.

4.3 Turbulence eddy frequency

Turbulence eddy frequency refers to frequency of eddies formed in the flow region which is expressed in terms of unit time period. Fig. 5 shows the distribution of turbulence eddy frequency for various injection inclinations. Increased turbulence eddy frequency is observed in the mixing region of vertical injection position. For 15° injection inclination eddy frequency rate is high in the mixing region and in the near wall region which is undesirable. The eddy frequency rate is comparatively low for 30° injection position.

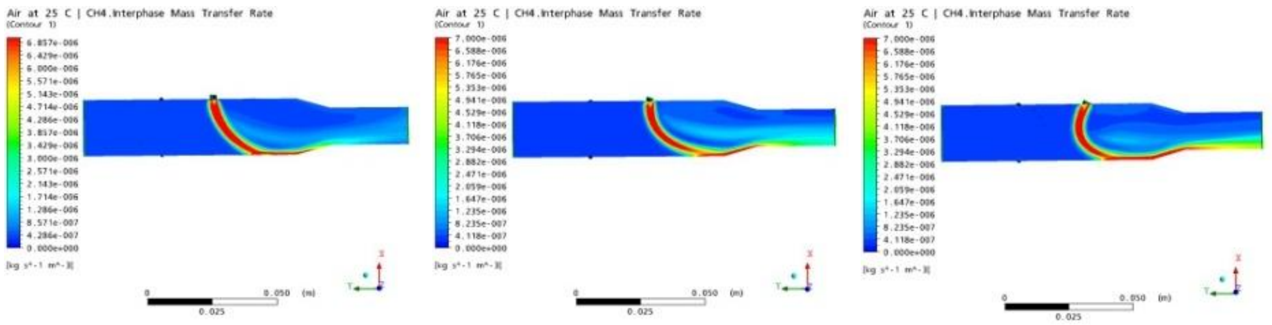
4.4 Turbulence kinetic energy

The turbulence kinetic energy (TKE) is characterised by measured root-mean-square (RMS) velocity fluctuations. The velocity fluctuation is directly proportional to intensity of turbulence in the flow field. Fig. 6 shows the TKE distribution for the various injection inclinations. Increased TKE is observed in the mixing regions for all injection inclination positions. This is quietly high for 30° inclination which is highly desirable. But increased level in the downstream is observed for 30° position causes the fluctuations in the flow which results in energy dissipation.



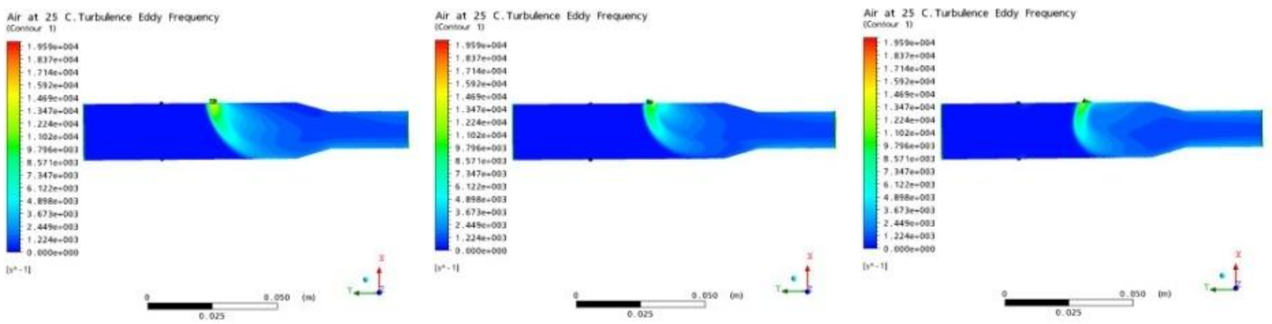
(a): Vertical position **(b): 15° inclination** **(c): 30° inclination**

Fig. 3: Distribution of CH_4 mass fraction



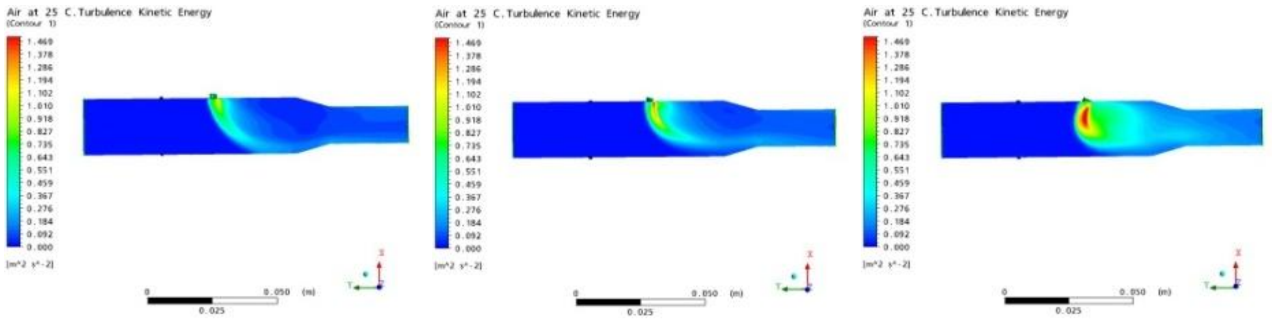
(a): Vertical position **(b): 15° inclination** **(c): 30° inclination**

Fig. 4: Interphase mass transfer rate



(a): Vertical position **(b): 15° inclination** **(c): 30° inclination**

Fig. 5: Turbulence eddy frequency



(a): Vertical position **(b): 15° inclination** **(c): 30° inclination**

Fig. 6: Turbulent kinetic energy variation

5. Conclusion

Based on the present analysis, the following conclusion has been drawn:

- Increased stoichiometric CH_4 mass fraction is observed in the mixing and outlet regions when the injection angle increases.
- Increased mass transfer rate is observed in the mixing region of injection angle 30°.
- The velocity fluctuation is directly proportional to intensity of turbulence in the flow field. The intensity of turbulence is comparatively high in the mixing region of injection angle 30° which is highly desirable for better mixing.

REFERENCES:

[1] R. Abu Bakar, Semin and A.R. Ismail. 2008. Application review of compressed natural gas as a sustainable alternative fuel in the internal combustion engines, *Proc. 1st Int. Conf. IET Brunei Darussalam Network*.

[2] C.S. Hua, Z. Yaacob and R. Mohsin. 2011. Effect of various manifold absolute pressures and air flow rates on throttle body injection mixer in compressed natural gas motorcycle, *J. Chemical and Natural Resources Engg.*, 1, 40-49.

[3] M.Y.T. Zamri. 2000. *Development of a 3D CFD Model to Investigate the Effect of the Mixing Quality on the CNG Diesel Engine Performance*, University of Southern Queensland.

[4] H.H. Geok and T.I. Ohamad. 2009. Experimental investigation of performance and emission of a sequential port injection natural gas engine, *European J. Scientific Research*, 30(2), 204-214.

[5] Semin, A. Idris and R. Abu Bakar. 2009. Effect of port injection CNG engine using injector nozzle multi holes on air-fuel mixing in combustion chamber, *European J. Scientific Research*, 34(2), 16-24.

[6] Semin, A.R. Ismail and R. Abu Bakar.. 2009. Gas fuel spray simulation of port injection compressed natural gas engine using injector nozzle multi holes, *European J. Scientific Research*, 29(2), 188-193.

[7] S. Kaleemuddin and G.A.P. Rao. 2009. Development of dual fuel single cylinder natural gas engine an analysis and experimental investigation for performance and emission, *American J. Applied Sciences*, 6(5), 929-936. https://doi.org/10.3844/ajassp.2009.929.936.

[8] A. Miura and F. Honjou. 2000. Further development of fuel consumption for heavy-duty CNG engine, *FISITA World Automotive Congress*, June 12-15.

[9] G. Barroso and A. Escher. 2005. Experimental and numerical investigations on HCCI combustion, *SAE Technical Paper*, 2005-24-038. https://doi.org/10.4271/2005-24-038.