

COMPARATIVE EVALUATION OF A TWO STROKE COMPRESSED NATURAL GAS MIXER DESIGN USING SIMULATION AND EXPERIMENTAL TECHNIQUES

Devarajan Ramasamy, Rosli Abu Bakar, Mohd Fadzil Rahim, M.M.Noor
Automotive Focus Group
University Malaysia Pahang
Locked Bag 12, 25000 Kuantan,
Pahang, Malaysia.
deva@ump.edu.my

ABSTRACT

Compressed Natural Gas (CNG) is a viable alternative fuel that is able to reduce tailpipe emission, most notably in two stroke engines. The excessive by-products of two-stroke engine combustion; normally due to inefficient combustion process is largely attributed to high particulate, carbon monoxide and hydrocarbon constituents. A prototype uniflow-type single-cylinder engine was equipped with a bi-fuel conversion system were used for the work. A dedicated mixer was also developed to meter the gaseous fuel through the engine intake system. It was designed to meet the air and fuel requirement similar to the gasoline counterpart. Modeling of the mixer was made to obtain optimum orifice diameter using three different sizes of 14, 16 and 18mm respectively. Here, flow simulations using a standard (Computational Fluid Dynamics) CFD software were extensively used and the predicted results were subsequently validated using a dedicated a flow test rig. Pressure drop across the venturi is an important parameter as it determines the actual fuel-air ratio in the actual engine. A good agreement of CFD outputs with that of the experimental outputs was recorded. This paper highlights the work, which leads to the use of the dedicated CNG fuelling system in a general-purpose gasoline two-stroke engine.

KEY WORDS

Compressed Natural Gas (CNG), Computational Fluid Dynamics (CFD), Two-Stroke, Mixer, Air Flow

1. Introduction

Internal combustion engine mixture preparation and utilization are mainly governed by fluid flow. A clear understanding of the fluid motion and dynamics processes are needed to improve the design of each of components involved. CFD can be used to give information on the properties of the fluid flow in the respective engine components. A two-stroke engine is designated for conversion into bi-fuel version to use compressed natural gas and gasoline respectively. A mixer was developed

based on several assumptions with the mixer size described by Maxwell [1]. The suction force predicted using a CFD technique must be tested to give a good correlation of the model before it can be used in an actual engine. The prototype was tested in a motored condition with the pressure readings obtained. The results are then compared with the simulation results. The motored flow profile must be thoroughly established before the CNG mixer can be used in the actual internal combustion engine.

2. Model Development

The actual model of the mixer was developed using the data of maximum airflow rate at the throat section is for the determination of its critical size [2]. The diameter was determined as 10mm for a small throat at sonic speeds. The size was then increased to 14mm, 16mm and 18mm using a set of interchangeable rings positioned at the throat of the mixer. The experiment on the various throat sizes was made, as a bigger venturi will facilitate the induction process especially when the engine operates in the high-speed region [1]. Here the variable parameters are the i) nozzle distance, ii) venturi diameter, iii) throttle opening and iv) different gas fuels as found by some previous researchers [3]. Figure 1 and 2 shows the geometrical features of the mixer and its critical throat inserts.

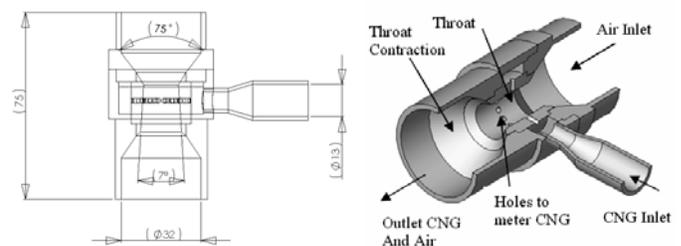


Figure 1. Technical Drawing and Section View of the Main Model

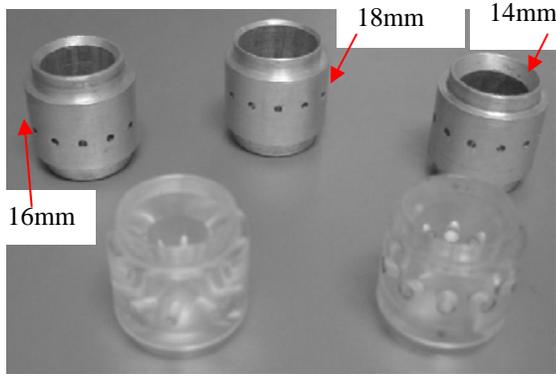


Figure 2. Fabricated Interchangeable Rings with Perspex Counterparts

3. Simulation and Validation of Model

Simulations were made on all the three ring inserts of the mixer. The simulation boundary conditions were considered as shown in Figure 3. There are three boundaries identified which are the air intake at two locations in the mixer and also the suction provided by the blower. The blower as shown in Figure 4 provides the necessary airflow required for the validation work. The flow rate was in range of 10 litre/min to 45 litre/min with an increment of 5 litre/min. The simulation was also carried performed for the different engine speeds to simulate the pressure drop across the throat of the mixer. This following Bernoulli equation was extensively used.

$$P_2 + \rho \frac{V_2^2}{2} = P_1 + \rho \frac{V_1^2}{2} \quad (1)$$

The pressure was noted to follow a quadratic trend with respect to the air velocity (engine speed). The CFD software utilizes the Navier Stokes equations to solve the flow behaviour [4].

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_k} (\rho u_k) = 0 \quad (2)$$

$$\frac{\partial \rho u_i}{\partial y} + \frac{\partial}{\partial x_k} (\rho u_i u_k - \tau_{ik}) + \frac{\partial P}{\partial x_i} = S_i \quad (3)$$

$$\frac{\partial (\rho E)}{\partial y} + \frac{\partial}{\partial x_k} ((\rho E + P)u_k + q_k - \tau_{ik}u_i) = S_k u_k + Q_H \quad (4)$$

Table 3.1

Properties of air obtained from Aneroid Barometer

Measured Parameter	Units
Pressure	101325 Pa
Temperature	29 °C

The simulation work was also based on the equations of i) mass momentum and ii) energy. Here, the fluid is assumed as air with properties shown in Table 3.1 used

throughout the work. The surface roughness was assumed to be 2 micrometer (based on the metrological work performed on its roughness). Additional assumptions are i) the wall of the mixer is assumed as adiabatic; ii) the outlet pressures are the total pressure (at ambient room conditions) as both the dynamic and static pressure are measured concurrently.

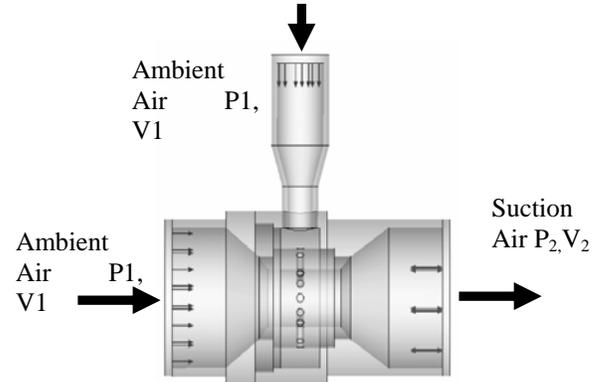


Figure 3. Boundary condition assumed in the mixer

4. Experimental Apparatus

Air induction in the rig is achieved by using the RCElec® blower (refer Figure. 4 (a)) and is controlled using Teco Inverter Speecon® model 7200MA. The variation of the resistance is through the knob control, with its frequency being displayed on the LCD Digital Operator on the Digital Display Unit. The Laminar Flow Element, (LFE) model 50MY15-5 Meriam Instrument®) is a meter that is used to measure the volumetric flow rate of air flowing through test section of the apparatus. The flow intensity of the LFE can be ranged from turbulent to laminar condition by means of air passing through small honeycomb passages with a high level of precision.

The pressure difference is measured using the TSI® VelociCalc pressure meter, shown in Figure 4(b). From here, both the readings (air flow rate and pressure difference) are obtained. The experiment is repeated for all the three ring inserts in the mixer. The effect of pressure difference against the blower air flowrate is as shown in Figure 5, 6 and 7 respectively.

5. Results

Figures 5, 6 and 7 indicate how the pressure increases with the increase in the blower speed. As the trend suggest, all the models provide similar trend which is increasing in quadratic. This is because in the Bernoulli equation pressure is quadratically proportional to air speed. CFD results show little variation with the experimentation. All the predicted reading depict a higher pressure trend than the one obtained from the experiment in the initial stage.

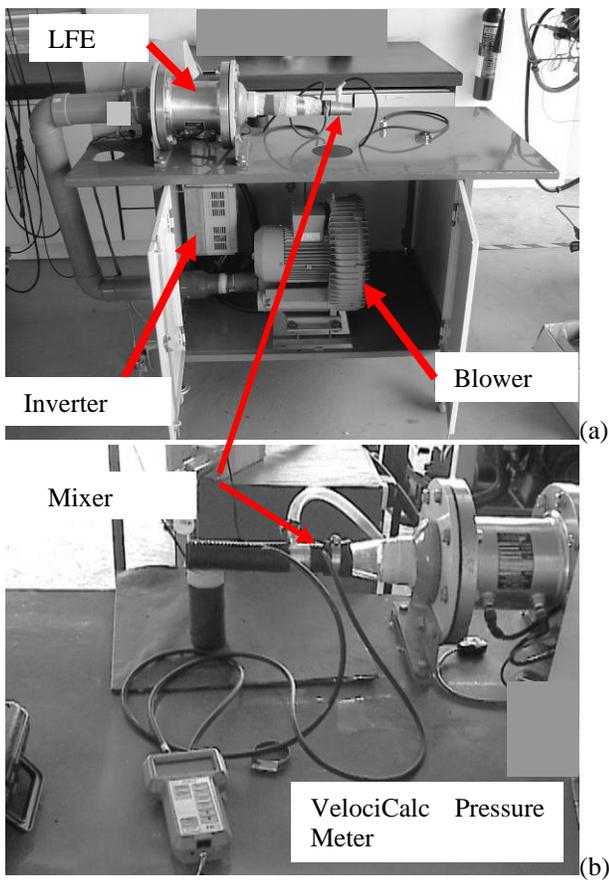


Figure 4. Flow Test Rig (a) and Pressure Difference Measurement (b)

A higher pressure value in the prediction is obtained as the simulation does not consider all the losses that might have occurred. Losses that might occur are probably due to pipe length and air viscosity changes however they are not considered as the variation is small. This was verified by assuming temperature being a constant as the average value observed for temperature change is too little. The small temperature variation gives very negligible change in pipe dimensions and also the fluid viscosity.

Once the value of 30 litre/min is exceeded, the simulated values exceeded those of the experimental values for all throat ring models. Ring 18mm has simulation readings more than experiment through out the experiment. From the results, the ring with diameter 16mm is assumed to be more accurate as the values correlates well with the experiment results. Flow readings at the beginning of the experiment are in close agreement with the predicted results but once reaching the maximum speed the values are wide apart.

In other hand, for 14mm and 18mm ring throat are slightly deviating from experiment. This may be due to the CFD software mesh is not very capable able to calculate flow near the wall and the air turbulence in the mixer correctly. As found by a researcher, CFD results are usually related to the number of cell might being insufficient, the boundary condition not appropriate

enough or the turbulence model does not match, which in this case is flow at high speeds Raghunathan [5].

In model with diameter 14mm and 16mm the magnitude of velocity is almost symmetric and there is suction velocity from the CNG inlet. Figure 8(a) and (b). Model with ring diameter 14mm shows the highest magnitude of suction. This model is hoped to give good performance to the engine at low speeds.

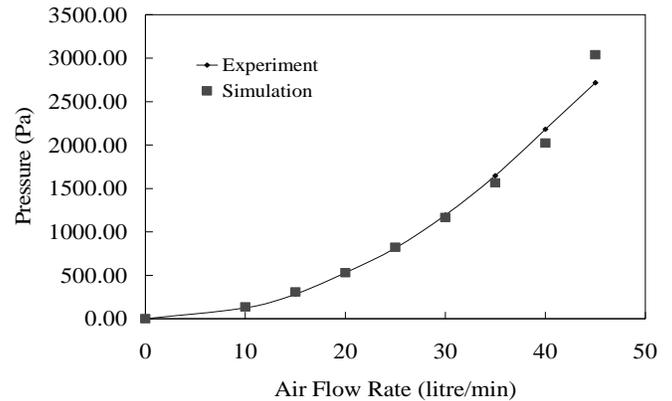


Figure 5. Effect of Air Flow on Pressure Difference for Mixer Throat Ring 14mm

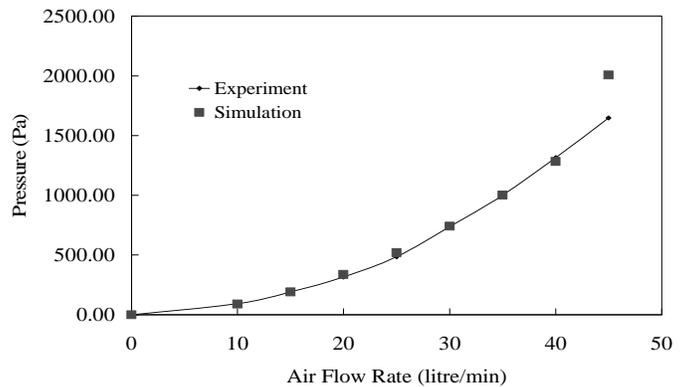


Figure 6. Effect of Air Flow on Pressure Difference for Mixer Throat Ring 16mm

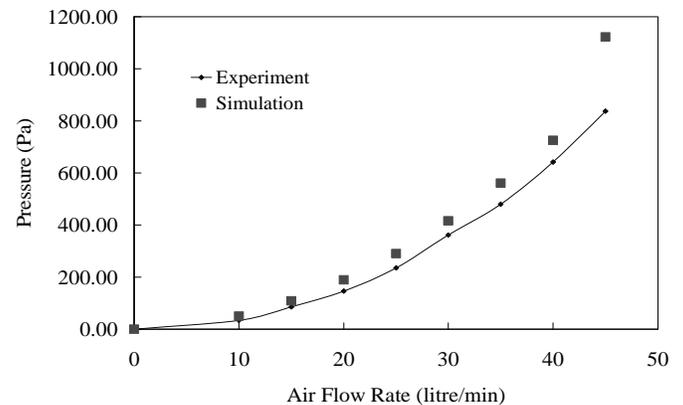


Figure 7. Effect of Air Flow on Pressure Difference for Mixer Throat Ring 18mm

By having a bigger suction the 14mm ring throat will ensure better fuel metering. In Figure 8(c) the flow is seen more turbulent in some areas of the mixer body, especially at the outlet 18mm. There is an unsymmetrical vector plot at the outlet area. This may be due to the small pressure drop that occurs in the model causing the flow to accumulate at the outlet as the velocity at the outlet is less compared to the two other models. The throat in this model is also not exactly following the diverging venturi angle. The less divergence angle causes sudden pressure drop and creates more turbulence. This cannot be avoided since it is due to the geometrical restriction of space in the model to create the venturi angle for the 18mm ring throat.

Based on the simulation results, model 16mm and 18mm will also be tested if the engine is to be used in high speeds. Testing with other rings will be carried out if model 14mm which is a small venturi cannot give a good performance at high engine speeds due to throat size restrictions.

6. Conclusion

The experimental technique is able to validate the pressure distribution predicted by CFD means on the effects of the three insert rings in the CNG mixer. The results are almost similar but not exact with regard to pressure distribution. The simulation exercise can be used to predict the amount of CNG consumed by the engine. The 14mm throat ring is found to be the best throat design for the CNG mixer as it gives the most suction but there are still some doubts of the throat functioning at high engine speeds. This will be cleared once the mixer is tested on a real engine.

Acknowledgement

The authors would like to express their sincere thanks to University Malaysia Pahang for sponsoring the conference fees.

References

- [1] Maxwell T.T. and Jones J.C., "Alternative Fuels: Emissions, Economics and Performance", USA Society of Automotive Engineers: SAE Inc., 1995.
- [2] Rosli Abu Bakar, Devarajan Ramasamy, Gan Leong Ming, "Design Of Compressed Natural Gas (CNG) Mixer Using Computational Fluid Dynamics", 2nd BSME-ASME International Conference on Thermal Engineering, 2004.
- [3] Bo Yan Xu, Mikio Furuyama, "Visualization of Natural Gas-Air Mixing Flow in the Mixer of a CNG

Vehicle", Technical Notes, JSAE Review 18 (1997) 57-82, 1996.

[4] CosmosTM, "Technical Reference Manual", COSMOSFloWorks Fundamentals., 2001.

[5] B.D. Raghunathan and R.G. Kenny, "CFD Simulation and Validation of the Flow within A Motored Two Stroke Engine", International Congress & Exposition Detroit, Michigan 1997,SAE 970359, 1997.

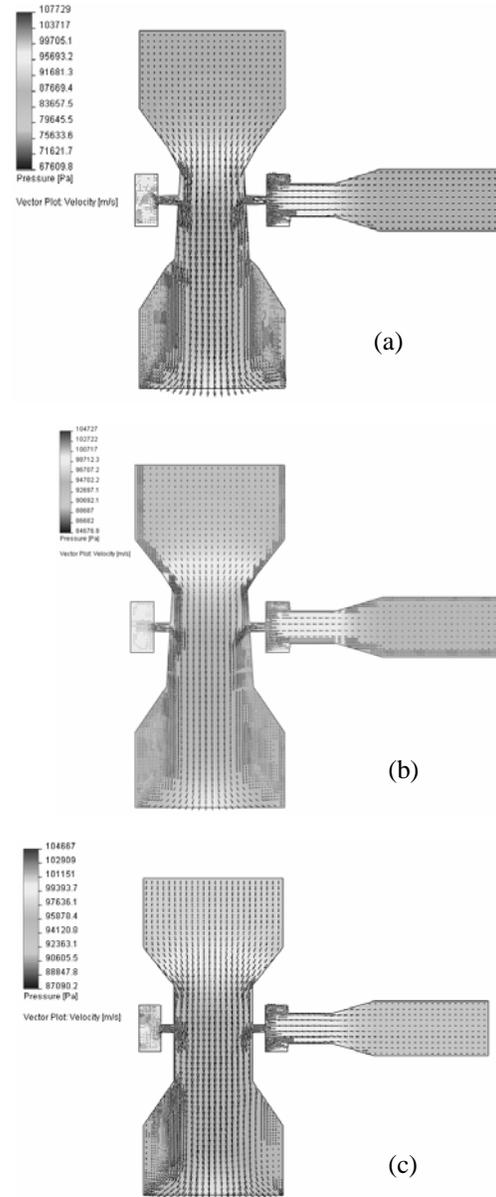


Figure 8 Vector and Pressure Difference Plot in Model (a) Throat Ring 14mm (b) Throat Ring 16mm (c) Throat Ring 18mm.