National Conference in Mechanical Engineering Research and Postgraduate Students (1st NCMER 2010)
26-27 MAY 2010, FKM Conference Hall, UMP, Kuantan, Pahang, Malaysia; pp. 592-599
ISBN: 978-967-5080-9501 (CD ROM); Editors: M.M. Rahman, M.M. Noor and K. Kadirgama

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

D. Ramasamy, S. Mahendran, Zamri Mohamed and Rosli A. Bakar

Faculty of Mechanical Engineering, Universiti Malaysia Pahang
26300 UMP, Kuantan, Pahang, Malaysia
Phone: +6016-7580946, Fax: +609-4242201
E-mail: deva@ump.edu.my

ABSTRACT

Natural gas is a viable alternative fuel that is able to reduce tailpipe emission, most notably in two stroke engines it is available in compressed form or Compressed Natural Gas (CNG) for easy storage. 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.

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

INTRODUCTION

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. The part of the bi-fuel kit that connects to the engine is the mixer. The design of the mixer is crucial to impart air-fuel mixture in a ratio for engine operation. In this study a CNG mixer was developed based on several assumptions of mixer sizes as described by (Maxwell, 1995). The suction force was predicted using a CFD technique. The results must be tested experimentally 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.

MODEL DEVELOPMENT

The 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 (Rosli Abu Bakar, 2004). 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 (Maxwell, 1995). 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 (Bo Yan Xu, 1996). Figure 1 and 2 shows the geometrical features of the mixer and its critical throat inserts.

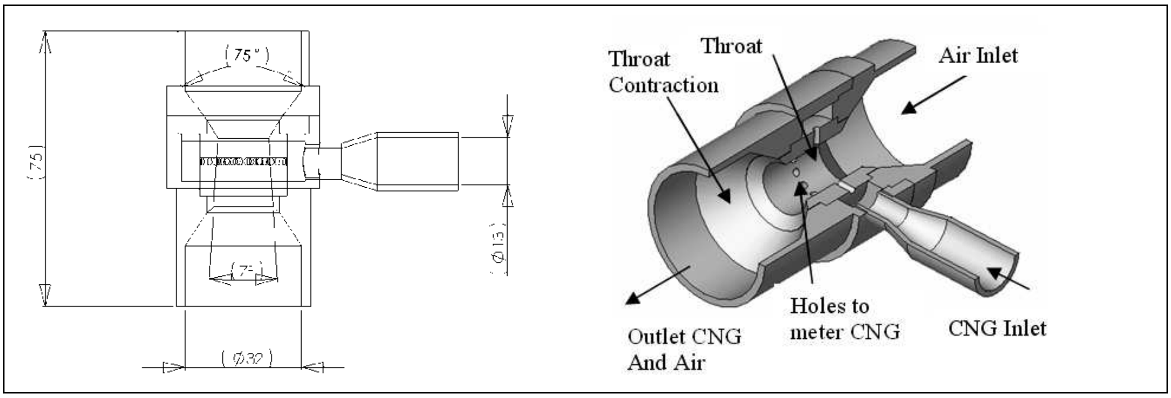

Figure 1: Drawing and The Section View Of The Main Model

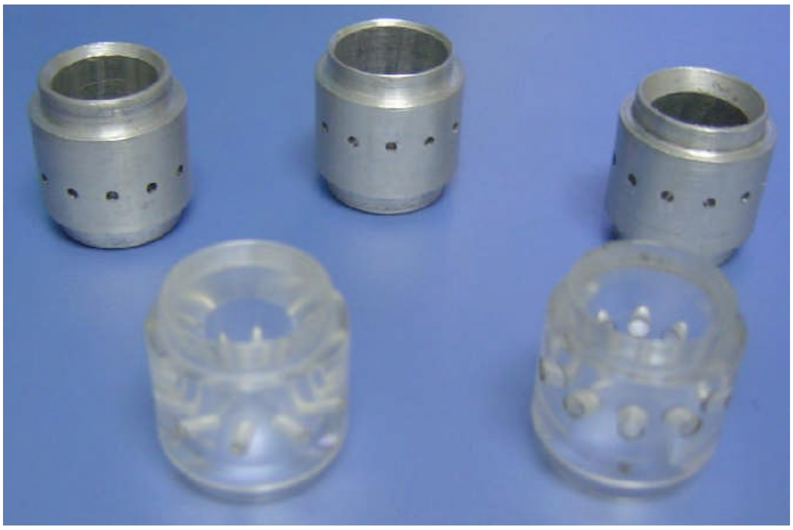

Figure 2: Fabricated Interchangeable Rings

SIMULATION AND VALIDATION OF MODEL

Simulations were made on all the three ring inserts of the mixer. The blower (see Figure 4) provides the necessary airflow required for the validation work. The flow rate was varied from 10 l/min to 45 l/min with an increment of 5 l/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.

$$P_2 + \rho \frac{V_2^2}{2} = P_1 + \rho \frac{V_1^2}{2} \tag{1}$$

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 (Cosmos, 2001).

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

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

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

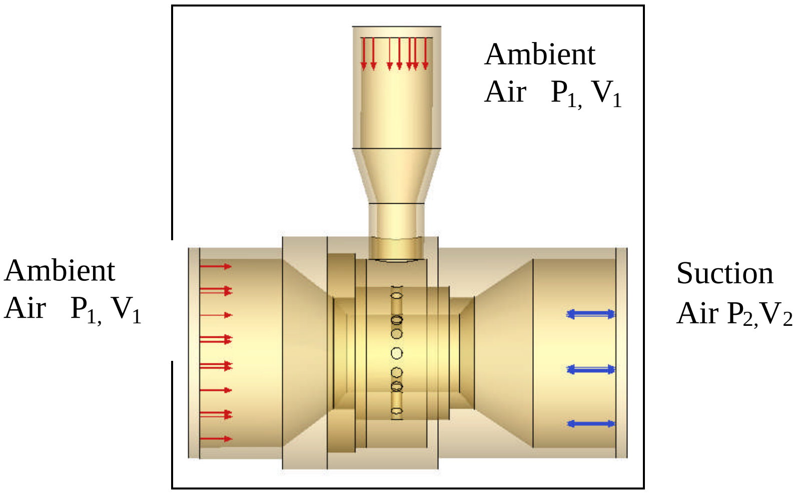

Figure 3: Boundary in the mixer

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.

Table 3.1: Properties of air obtained from Aneroid Barometer.

Pressure, Pa	101325 Pa
Temperature, °C	29 °C

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.

EXPERIMENTAL APPARATUS

Air induction in the rig is achieved by using the *RCelec*® blower (refer Fig. 4) 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 calibrate for the volumetric flow rate of air flowing through test section of the apparatus. The flow intensity can be varied from turbulent to laminar condition by using a small honeycomb passage with a high level of precision easily achived.

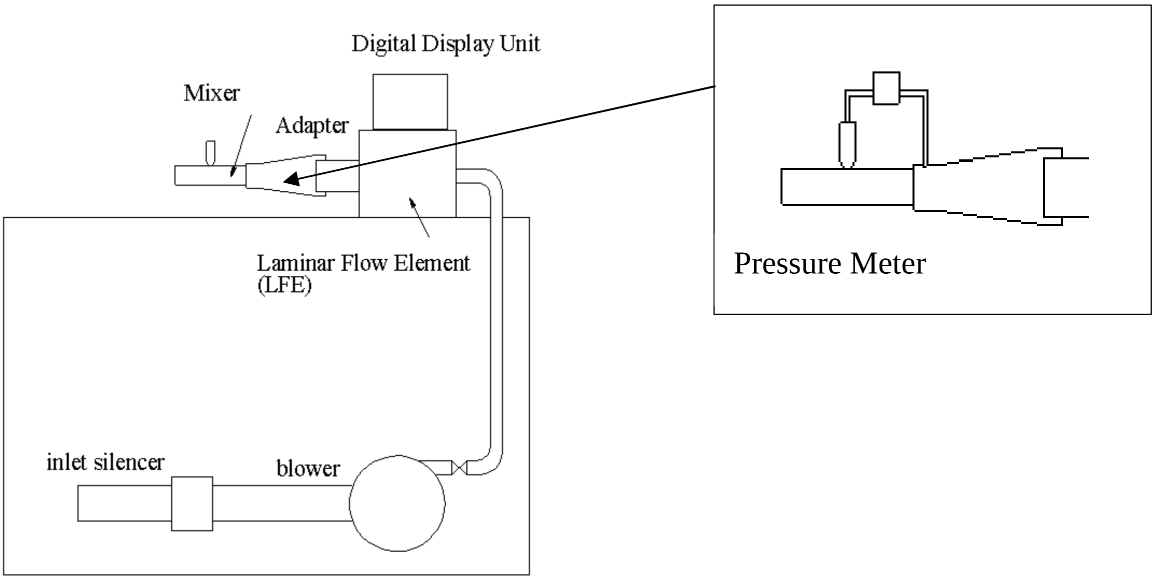

Figure 4: Schematic of Flow Test Rig for CFD Validation

The pressure difference is measured using the TSI® *VelociCalc* pressure meter, shown in Figure 3. From here, both the readings (mass flow rate and pressure difference) are obtained. The experiment is repeated for all the three ring inserts in the mixer. The pressure readings are plotted against the blower speed. The plot is compared with simulated data of the same speeds, as in shown in Figure 4, 5 and 6 respectively.

RESULTS

The graphs below 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 depicted a higher pressure trend than the one obtained from the experiment in the initial stage. A higher pressure value in the prediction is obtained as the simulation does not consider all the losses that might have occurred. Losses due to pipe length and air viscosity changes are not considered. Air viscosity will change as the temperature during testing is also changing. The temperature is assumed constant as the value changing is too small. Once the value of 30 lpm is exceeded, the predicted values exceeded those of the experimental values. The CFD software mesh setting of the flow might not be enough. A good mesh setting might solve this problem but the computational time will increase. Only ring 18mm has prediction readings more than experiment through out the experiment.

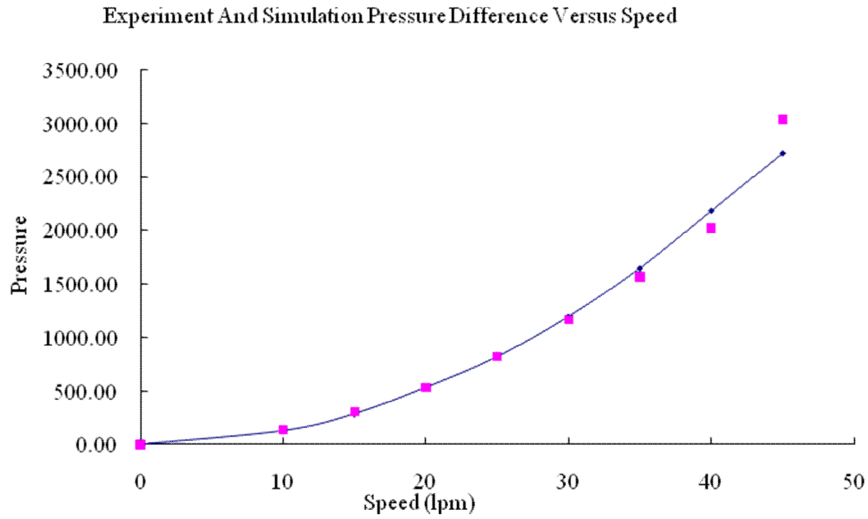

Figure 5: Mixer Ring 14 mm

From the readings, the ring with diameter 16 mm 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. This may be due to the CFD software mesh not able to calculate flow near the wall and the air turbulence in the mixer correctly. The number of cell might be insufficient, the boundary condition might be inappropriate or the turbulence model does not match the flow at high speeds as found by (B.D. Raghunathan, 1997).

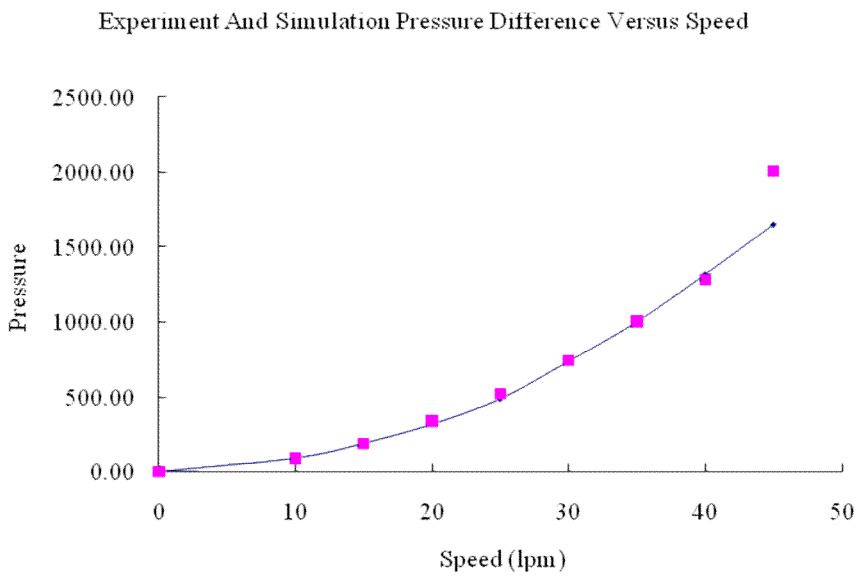

Figure 6: Mixer Ring 16 mm

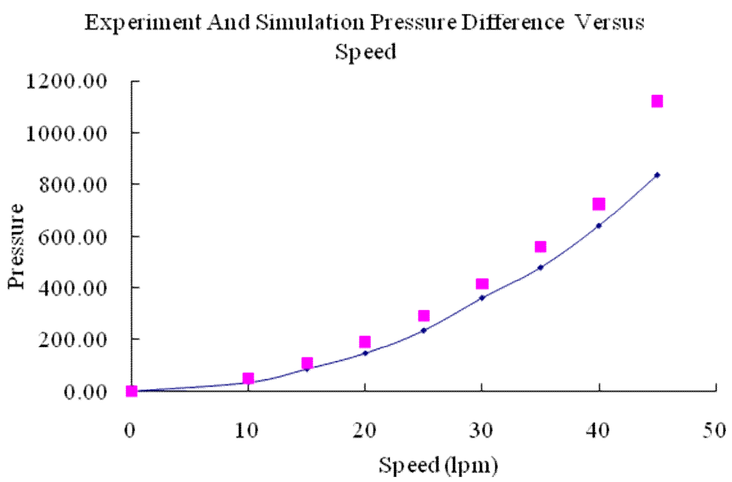

Figure 7: Mixer Ring 18 mm

In Figure 8(c) the flow is seen more turbulent in some areas of the mixer body, especially at the outlet 18mm. This may be due to the small pressure drop that occurs in the model causing the flow to accumulate as the velocity is less compared to the two models. The throat in this model is not following the venturi angle for outlet flow. This is due to the lack of space in the model to create the venturi angle. 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. A bigger suction will ensure better fuel metering at all engine speed.

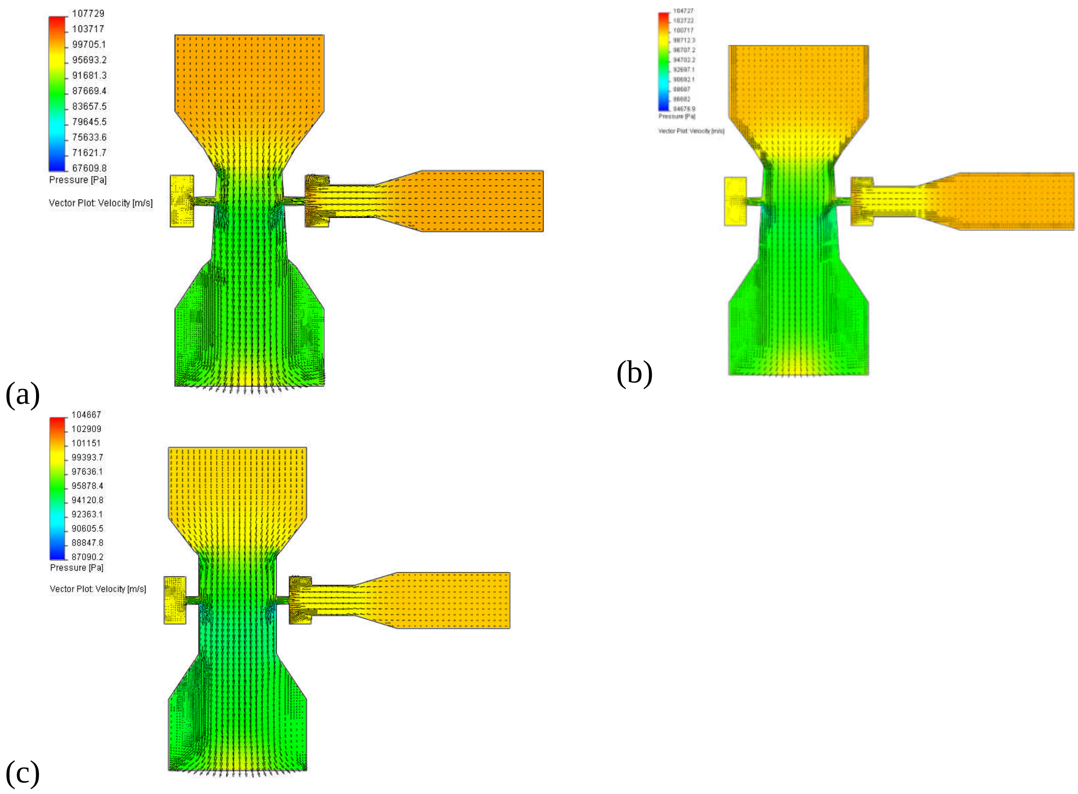

Figure 8: Simulation Results Of Pressure In Model (a) Ring 14mm (b) Ring 16 mm (c) Ring 18 mm.

Model 16 mm and 18mm will also be tested if the engine is to be used in high speeds. This is done in case model 14mm which is a small venturi cannot give a good performance at high engine speeds due to throat size restrictions.

CONCLUSIONS

The following conclusions are hereby made:

1. 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.
2. The results are almost similar but not exact with regard to pressure distribution.
3. The simulation exercise can be used to predict the amount of CNG consumed by the engine.

ACKNOWLEDGMENT

The authors would like to express their gratitude to Universiti Malaysia Pahang for sponsoring this paper under RDU090366.

REFERENCES

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

Bo Yan Xu, Mikio Furuyama, 1996, *Visualization of Natural Gas-Air Mixing Flow in the Mixer of a CNG Vehicle*, Technical Notes, JSAE Review 18 (1997) 57-82.

Cosmos^{TM} , 2001, *Technical Reference*, COSMOSFloWorks Fundamentals.,

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

Rosli Abu Bakar, Devarajan Ramasamy, Gan Leong Ming, 2004, Design Of Compressed Natural Gas (CNG) Mixer Using Computational Fluid Dynamics, *2nd BSME-ASME International Conference on Thermal Engineering 2004.*

SolidWorks 2001, pg 1-5, 2001, Dassault Systemes, USA.

Nomenclature

C_p	specific heat at constant pressure $J.kg^{-1}.K^{-1}$
g	acceleration due to gravity $m.s^{-2}$
k	thermal conductivity $J.m^{-1}.K^{-1}$
P	Pressure
t	time s
u	velocity of the fluid in the x-direction $m.s^{-1}$
V	velocity of $m.s^{-1}$

Greek symbols

μ	coefficient of viscosity Pa.s
ν	kinematic viscosity $m^2.s^{-1}$
ρ	density of the fluid $kg.m^{-3}$
τ	viscous shear tensor
Q_H	heat source or sink per unit volume, J
S_i	mass distributed external force per unit mass, N/kg