COMPUTATIONAL STUDIES OF FUEL AND AIR MIXING CHARACTERISTICS OF A LOW PRESSURE DOMESTIC GAS APPLIANCE

Azeman Mustafa and Noor Shawal Nasri

Department of Gas Engineering
Faculty of Chemical and Natural Resources Engineering
Universiti Teknologi Malaysia
81310 Johor, Malaysia
Email : azeman@fkkksa.utm.my

Abstract

Computational fluid dynamics (CFD) simulation has been carried out to study the fuel and air mixing characteristics of a low pressure domestic gas appliance. Three types of liquefied petroleum gases (LPG), namely pure propane, pure butane and a mixture (by weight) of 30% propane and 70% butane, were simulated at a fuel pressure of 300 mm WG. The CFD results show that for the same fuel supply pressure and gas orifice size, the mass flow rate of LPG fuel discharged into a burner was proportionate to its specific gravity. However, as compared to butane and LPG mixture, the propane fuel discharge velocity was the highest due to its lowest specific gravity. This subsequently produced the most negative static pressure values at a burner mixing tube throat and hence allowing the largest amount of primary air to be induced into an appliance. The amount of primary air required to initiate combustion by propane, LPG mixture and butane fuels was predicted to be approximately 72%, 61% and 54% of stoichiometric requirement, respectively. These values are in good agreement with a typical range of primary air requirement for this type of atmospheric burner. The mixing rates of propane-air system were predicted to be much faster than that of LPG mixture-air and butane-air systems. However, for all three types of LPG fuel, the fuel and air mixing was found to be almost complete in the second mixing tube. In addition, the average mass fraction of fuel at a burner input was predicted to be almost identical to that of burner outlet, thus demonstrating a fully-premixed capability of this type of atmospheric gas appliance.

Keywords

Low pressure gas appliance, air-fuel mixing, primary air requirement

Introduction

Energy is the key ingredient to any activity. Adequacy of energy supply is important for any economy to prosper. However, efficient production and use of energy is equally important for sustaining continuous economic production and ensuring healthy living environment. This is because inefficient generation and consumption of energy not only contributes to economic losses but also produces some undesirable impacts on the environment, such as global warming, acid rain, ozone depletion and health effects.

Like in any developing nation, energy consumption per capita in Malaysia is still low but is expanding at a rapid rate in tandem with economic development. In addition, energy intensity with respect to gross domestic product had shown an increasing trend historically. Twenty years ago, wood fuels were used as the main domestic energy source but as the country progressively develops , more efficient burning and cleaner fuels such natural gas and liquefied petroleum gases (LPG), have steadily replaced wood. In domestic sectors, LPG is still more preferred to natural gas as it is available in easily refillable and moveable storage tanks (12, 14 or 50 kg cylindrical tank). Furthermore, with the completion of Peninsular Gas Utilisation (PGU) project, natural gas which has already been used in power generation, petrochemical , transportation and commercial sectors can be readily available to those residing near the pipeline.

LPG and also natural gas used as domestic fuels usually operate at low gas pressure (i.e. 300 mm H_2O) using an atmospheric-type gas appliance. This is because this type of gas appliance is simple, easy and safe to operate as it does not require any mechanical and electrical parts. However, as nowadays this type of domestic burner is commercially available in various models, nobody can tell whether they are designed to burn fuel efficiently and safely. Combustion efficiency and stability is a very important term for any burner design as it indicates the degree of completeness and fuel being burned and the stability of flame sustained on the burner ports. Efficient burner operation produces no or very little safety-associated problems such as flame blow-off and flashback and also excessive emissions of undesired combustion products, in particular carbon monoxide.

There are two important considerations governing efficient and safe operation of an atmospheric-type gas burner [Steiner (1946) , Harris and South (1978)]. The first consideration is a proportion of primary air required to initiate fuel burning or combustion. This is very much dependent on , as will be discussed later, how combustion air is induced and mixed with fuel in a mixing tube before they are burned. The second consideration is a method required to ensure that upon successful ignition, flames will be self-propagating and self-sustaining on a burner head. This is important as unstable flame, as previously mentioned, will lead to problems such as flashback or backfiring, pop extinction and flame lifting. Thus, the burner head design which comprises of different types of burner port and methods to stabilize flames are of parameters of interest.

Burner aerodynamic flow fields fuel and air mixing characteristics is a necessity for the improvement of fuel burning efficiency and pollution reduction in many combustion devices. In the past, the combustion designer has traditionally been forced to rely heavily on experience, experimental data and empirical expressions. This traditional technique is capable of providing important information about the nature of the flow and combustion process but it would soon prove both very expensive and incapable of assimilating the vast amount of design information needed in future's development and evaluation of more efficient and pollution-free combustion devices. Fortunately, the rapidly developing computational fluid dynamic (CFD) capability makes possible for computer modeling to reduce the amount of costly and time-consuming experimental procedures The technique is very powerful and spans a wide range of industrial and non-industrial application areas. CFD techniques involve the numerical solution of the fundamental equations of conservation of mass, momentum, energy and individual species, closed by some turbulence model, on a computational grid or mesh fitted to the geometry of interest. Detailed measurements of velocity, temperature, species concentration etc. are required for formulation and verification of computer models. There are several commercially available CFD codes such as FLUENT, PCOC, FLOW3D etc. that can be used to predict the isothermal or reacting characteristics of a flow system of interest.

In combustion related-areas CFD applications not only contribute to the potential cost and time savings but also provide additional insight into complex problems that is not feasibly accomplished by analytical solutions or may be too costly and time consuming to pursue experimentally [Sturgess and Syred, 1983]. The availability of affordable high performance computing hardware and the introduction of user-friendly interfaces have led to a recent upsurge of interest and CFD is poised to make an entry into the wider industrial community in the next millennium. The use of computational fluid dynamic (CFD) to predict internal and external flows in gas turbine system has risen dramatically in the past decade. Most CFD investigations of gas turbines combustors have been concentrated on velocity and turbulence of swirling flows [Isaac, 1992; Robert and Larry, 1994]. In recent years, attempts have also been made to include the CFD predictions of combustion emissions, in particular NOx [Amin et al., 1994].

Present CFD approach

The present work is carried out with the objective of gaining detailed knowledge of aerodynamics and air/fuel mixing characteristics of current low pressure domestic gas appliances by performing CFD studies using a commercially available FLUENT code. As there are so many models of gas appliances currently available in the market, some of which do not even have approved certification from an authorized body such as SIRIM, no one can tell whether they are designed to operate efficiently and safely. Hence, it is the intention of this CFD study to provide some detailed information that can be used as guidelines by burner manufacturers. The present CFD investigation are divided into two parts; the first part involves isothermal CFD studies investigating the flow aerodynamics and fuel and air mixing characteristics of atmospheric-type gas appliances with different mixing tube design specifications. Whilst, the second part investigates combustion characteristics of different burner head design specifications using air / fuel data obtained from previous cold studies. This paper partly reports the isothermal CFD results of the fuel air mixing characteristics of the base design gas appliances.

The accuracy of the CFD model is quite difficult to be validated as at present, there are yet experimental data available. Nevertheless, the gross accuracy of the present CFD results can be approximated by comparing with the recommended burner design guidelines by American Gas Engineer Institute.

The CFD model with a three-dimensional unstructured mesh generation employing GAMBIT / FLUENT 6.0 was geometrically identical to a selected low pressure domestic gas appliance, as schematically illustrated in Figure 1.

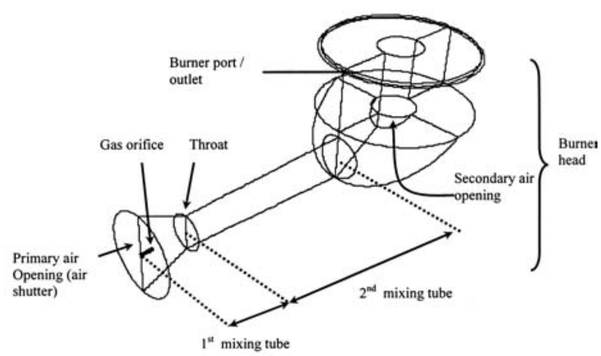

Figure 1 – A schematic diagram of low pressure gas appliance

The model design specifications are given Table 1. The governing equations for isothermal flows comprised of the Navier-Stoke equations and equations for the energy, species concentration and equations of state. The very low intensity turbulence mixing was assumed using the unmodified constants two-equation RNG κ–ε turbulent model. The governing equations were discretised by a finite difference method and integrated over the computational cells into which the domain has been divided. The integration was fully implicit and all the components were calculated at the nodal points of the cell. The simulation was carried out using the Power-law Interpolation discretisation scheme.

Design specifications	
Primary air opening radius = 17 mm Throat radius = 7.5 mm 2nd tube end radius = 10 mm Length of 1st tube = 20 mm Length of 2nd tube = 80 mm Gas orifice diameter = 1 mm Gas orifice insertion = 5 mm from primary air opening	Throat area = 8.83×10^{-05} m^2 Burner port =2.2133×10^{-04} m^2 Primary air opening = 4.5333×10^{-04} m^2 Gas orifice = 4.278×10^{-07} m^2 $A_{throat} / A_{port} \sim 0.4$ $A_{shutter} / A_{port} \sim 2.05$ 1st tube ~ 1.3 x throat diameter 2nd tube ~ 5.3 x throat diameter

Table 1 : CFD model design specifications

Summary of selected CFD results

Fuel type	Fuel flow rate		Primary air flow rate		Primary air % of stoichiomteric requirement
	(kg/sec)	(m/sec)	(kg/sec)	(m/sec)	
100% Propane	3.98e-05	52.23	0.000451	0.9668	71.59
100% Butane	4.56e-05	45.59	0.000383	0.8230	53.89
30% w/w propane & 70% w/w Butane	4.36e-05	47.69	0.000414	0.8902	60.63

Table 2 : Fuel and primary air mass flow predictions

Fuel type	Static pressure (Pa) at			Pressure drop (ΔP) between air opening and throat
	Air opening	Throat	Burner port	(Pa)
100% Propane	- 0.5956	- 8.1354	0.6335	7.5398
100% Butane	- 0.4321	- 5.5356	0.4190	5.1035
30% w/w propane & 70% w/w Butane	- 0.5052	- 6.7119	0.4864	6.2067

Table 3 : Static pressure predictions

Fuel type	Average mass fraction of fuel at burner inlet	Average mass fraction of fuel at burner outlet
100% Propane	0.08109	0.08127
100% Butane	0.1065	0.1064
30% w/w propane & 70% w/w Butane	0.1018	0.09524

Table 4 : Fuel mass fraction predictions

Discussion

One of the main problems encountered and the most difficult consideration during the CFD simulation was whether the fuel and air mixing takes place in a laminar or turbulent mode. With a fuel pressure of 2500 Pa, the corresponding discharge velocity (from a gas orifice) as shown in Table 2, was well within a turbulent region. In contrast, the primary air velocity was, as also shown in Table 2, in a laminar region, thus creating a laminar/turbulent flow system. Since the model simulated was a low pressure domestic gas appliance, a laminar mixing mode was initially assumed. However, the partially converged solution showed that there was virtually no fuel and air mixing taking place. However, with an assumption of a very low intensity of turbulent mixing mode, the CFD results demonstrated reasonable fuel and air proportion relationships and as well as mixing profiles. The CFD turbulent energy predictions were also very small indicating a very low turbulent mixing mode. This assumption was also considered in the work of Soo and Woo [2002].

Three case studies involving three types of liquefied petroleum fuels (LPG), namely pure propane, pure butane and a mixture of 30% w/w propane and 70% w/w butane were computationally investigated using a low pressure domestic gas appliance model with design specifications, as listed in Table 1. The CFD parameters of interest are amounts of fuel and primary air, some flow-associated parameters such as static pressure, velocity magnitude and turbulent energy, and also air and fuel mixing profiles inside an appliance.

Table 2 compares the fuel and air mass flow rate predictions of different LPG composition. With the same gas orifice diameter (i.e. 1 mm) and fuel pressure (i.e. 2500 Pa), the fuel specific gravity plays an important role in determining the flow rate of fuel admitted into an appliance and hence the rating of burner. The highest inlet mass flow rate and hence burner energy input was predicted when using pure butane, followed by 30%/70% by weight propane/butane mixture and pure propane. Nevertheless, due to its lowest specific gravity among the three LPG fuels investigated, pure propane produced the highest inlet fuel velocity, followed by 30%/70% by weight propane/butane mixture and pure butane.

Table 3 shows that the highest inlet pure propane velocity prediction contributed to the most negative values of static pressure at an (primary) air opening and as well as at a throat connecting two burner mixing tubes, as schematically illustrated in Figure 1. The highest vacuum or suction pressure created at the air opening and as well as the highest pressure drop occurred in the first mixing tube allowed the highest amount of (primary) air to be induced into an appliance. From Table 2, the predicted amount of primary combustion air induced when an air shutter is fully opened, for pure propane, 30%/70% by weight propane/butane mixture and pure butane were approximately 72%, 54% and 61%, respectively. These values are well within a normal recommended range of primary air requirement for a low pressure gas appliance.

Figure 2 shows that the maximum velocity magnitude took place at the center-line region of the first mixing tube, a region in which the fuel was discharged from a gas orifice. The fastest axial velocity developed with propane, followed by LPG mixture and butane fuels in the second mixing tube, as shown at X=20 mm, 40 mm and 80 mm after the burner throat, indicates the magnitude of positive head pressure developed in the burner head section. Table 3 shows that the burner port pressure predictions of propane, LPG and butane fuels were approximately 0.6335 Pa, 0.4864 Pa and 0.4190 Pa, respectively.

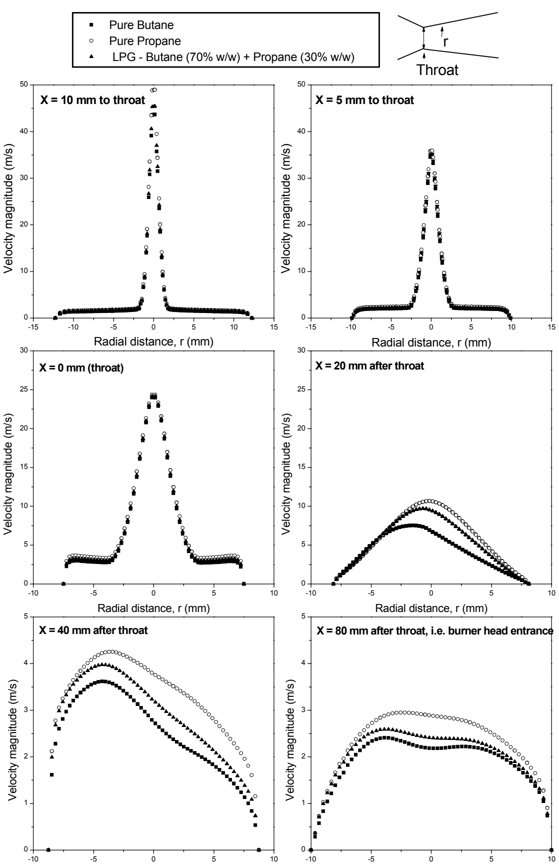

Figure 2 – Axial velocity predictions

One main problem anticipated from the combined effects of the leanest combustible mixture, as indicated by the fuel mass fraction predictions in Table 4, and the highest pressure drop (relative to atmospheric ambient) occurring at the burner port with propane fuel is that the propane-air mixture is much more difficult to burn or may encounter flame stability problems such as flame lifting or blow-off. One possible solution for this problem is that an amount of primary air should be reduced by partially closing an air shutter, thus allowing a more 'fuel-rich' propane-air mixture. With much more 'fuel-rich mixture', butane-air combustion is expected to be more stable than LPG-air combustion. Nevertheless, the flame retention mechanism on burner ports is of another very important design parameter need to be considered. The effect of various flame retention mechanisms on combustion stability and characteristics will be a subject of future CFD work.

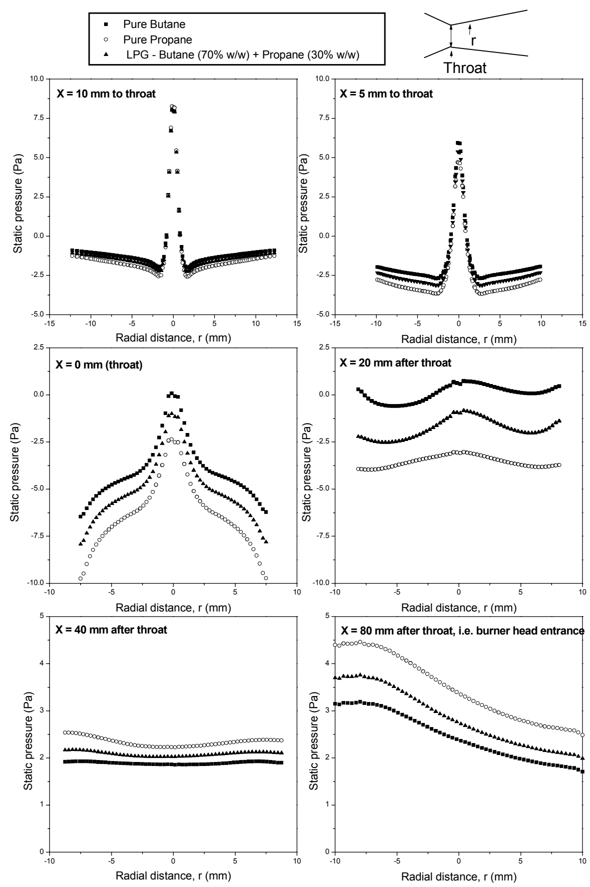

Figure 3 – Static pressure predictions

Figure 3 compares the radial static pressure predictions inside the burner mixing tubes for three LPG fuels. As compared to the velocity profiles in Figure 2, the static pressure profiles provide more distinctive flow dynamics especially in a region of almost identical axial velocity, i.e. first mixing tube. The most negative static pressure predicted inside the first mixing tube with propane fuel provided the highest suction or vacuum pressure and hence the largest amount of primary air to be induced into an appliance. In the second mixing tube, the propane fuel simulation once again demonstrates the fastest increase in static pressure leading to the highest burner port head pressure, as shown in Table 3.

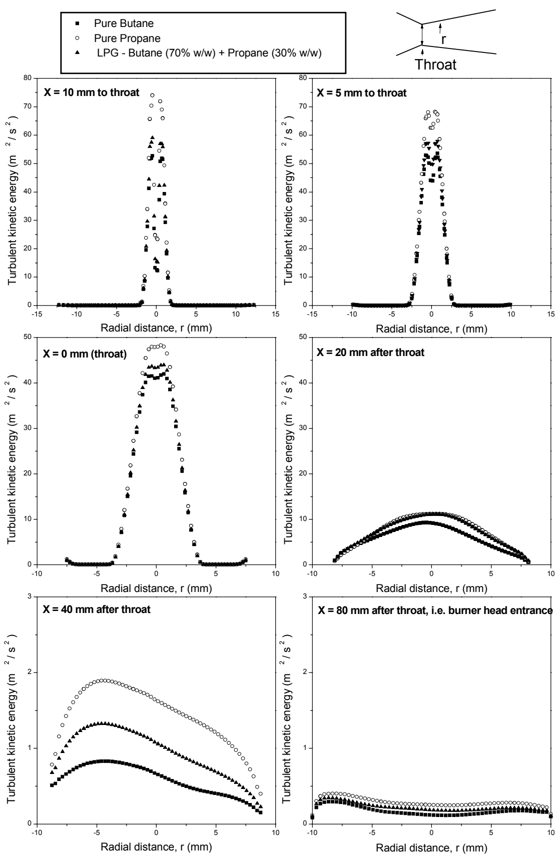

Figure 4 – Turbulent energy predictions

Figure 4 compares the radial turbulent energy predictions inside the burner mixing tubes for three LPG fuels. As expected, the fuel and air mixing took place in a very low turbulent environment. The most turbulent region was predicted to be in the first mixing tube where most fuel and air mixed. This is demonstrated by the fastest peak fuel richness reduction in Figure 5. At a burner mixing throat, the peak fuel richness at the center region was reduced to 35%-45% , thus indicating the fuel was already 55%-65% mixed with air. Upon passing through the burner mixing throat, the flow was rapidly expanded resulting in very rigorous fuel and air mixing, as indicated by a sharp decrease in peak fuel richness (from up 45% at throat to approximately 15% at X=20 mm after throat). In the second mixing tube , the CFD predictions show a steady decrease turbulent energy with tube length thus indicating less fuel and air mixing activities towards the burner head. Figure 5 shows the fuel was almost completely and uniformly mixed with air upon entering a burner head (i.e. X=80 mm). When comparing the fuel mass fraction predicted at inlet conditions with that of burner outlet or port , as shown in Table 4, it is found both values are almost identical. Thus , the CFD simulation shows that this type of an atmospheric gas appliance can achieve a fully-premixed mode.

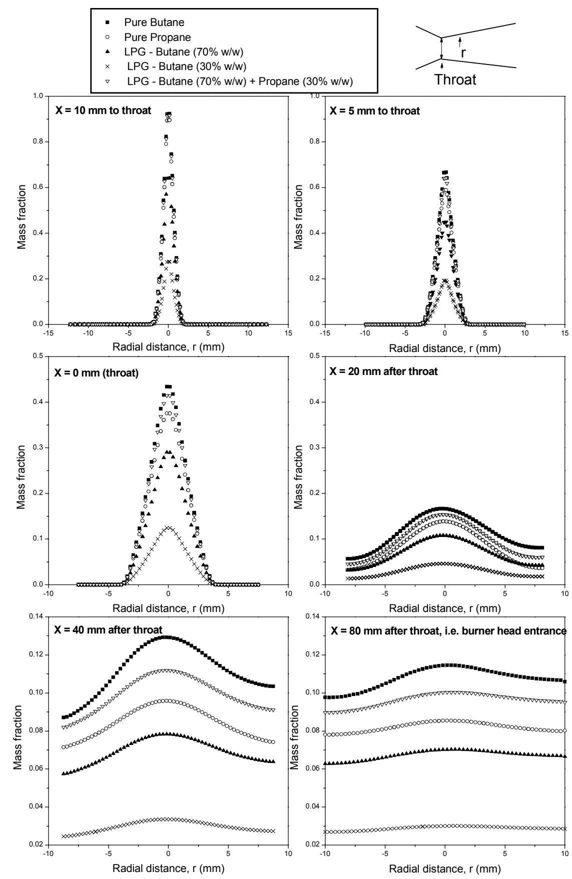

Figure 5 – Fuel mass fraction predictions

Conclusions

The preliminary CFD results show that CFD techniques are capable of providing detailed and yet quite reliable information about mixing properties of fuel and air inside a simple gas burner system such as a low pressure domestic gas appliance. The present CFD predictions of isothermal mixing clearly show that for the same burner design specifications, the quantity of primary air requirement and as well as the uniformity of fuel and air mixing very much depend on the gaseous fuel composition. A domestic gas appliance operation with lower molecular weight LPG fuel such as pure propane was predicted to induce primary air closer to its stoichiometric requirement. In addition, the fuel and air was predicted to attain complete mixing well inside a burner mixing tube before they burned.

Acknowledgments

The authors would like to thank Universiti Teknologi Malaysia for UPP grant (vot 71830).

References

Amin, E.M., Andrews G.E., Escott, N.H., Kim, M.N and Pourkashanian, M. 1994. Thermal and Prompt NOx 3D CFD predictions for An Ultra-low NOx Lean Radial Swirl Gas Turbine Combustion Primary Zone. ASME IGTI Int. Gas Turbine and Aero Engine Congress.

Isaac, K.M. 1992. A Computational Study of Gas Turbine Combustion Chamber Flow Field, *International Journal of Turbo and Jet Engines* 9: 285-299.

Soo-Jin, J. and Woo-Seung, K. 2002. A Study pn the Optimal Monolith Combination for Improving Flow Uniformity and Warm-Up Performance of an Auto-Catalyst. Chemical and Processing 00:1-17.

Steiner K. 1946. Fuels and Fuel Burners. McGraw-Hill Book Company Inc. , New York.

Sturgess, G. J. and Syred, S. A. 1983. Importance of Inlet Conditions for Numerical Simulation of Combustor Flows. AIAA paper 83-1263.