COMBUSTION STUDIES OF BLENDED FUEL OIL AS AN ALTERNATIVE FUEL FOR INDUSTRIAL GAS TURBINES (AFTUR) AND CONTROLLING NOX AND SOOT EMISSIONS

Ahmad Hussain[1,*], Farid Nasir Ani[1], Azeman Mustafa[2], Zainul Asri Mamat[3], Zulfadhly Zardi[3] and Abdul Aziz[4]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai 81310, Johor, Malaysia

[2]Faculty of Chemical and Natural Resources Engineering, Universiti Teknologi Malaysia, Skudai 81310, Johor, Malaysia

[3]TNB Research Sdn. Bhd., No. 1 Lorong Air Hitam, 43000 Kajang, Selangor, Malaysia.

[4]Prai Power Station, TNB Generation Sdn. Bhd., 13609 Prai, Penang, Malaysia

1.0 INTRODUCTION

Global interest in clean power generation has driven continued improvement of power systems. Improvement efforts focus on reducing emissions, improving efficiency and lowering costs without sacrificing reliability. Advances are occurring on all three fronts, but progress is usually achieved on a single-composition fuel. Thus, there is a need to consider fuel composition and fuel property variables on advanced power generation systems.

Current gas turbine systems that are capable of burning fuels are normally developed for a single specific fuel (such as natural gas or domestic fuel oil) and use conventional diffusion flame technology with relatively high levels of NOx and partially unburned species emissions. **(Adoune et. al 2001)**

Extensive research work is being done in atomization and spray characteristics of liquid fuels, the combustion and emission characteristics of the selected alternative fuel flames, the sooting tendency and NOx emission properties of the fuels under various operation conditions, and the adaptability of existing conventionally fuelled combustors for use with the selected alternative fuels **(Sharma et. al 2004).**

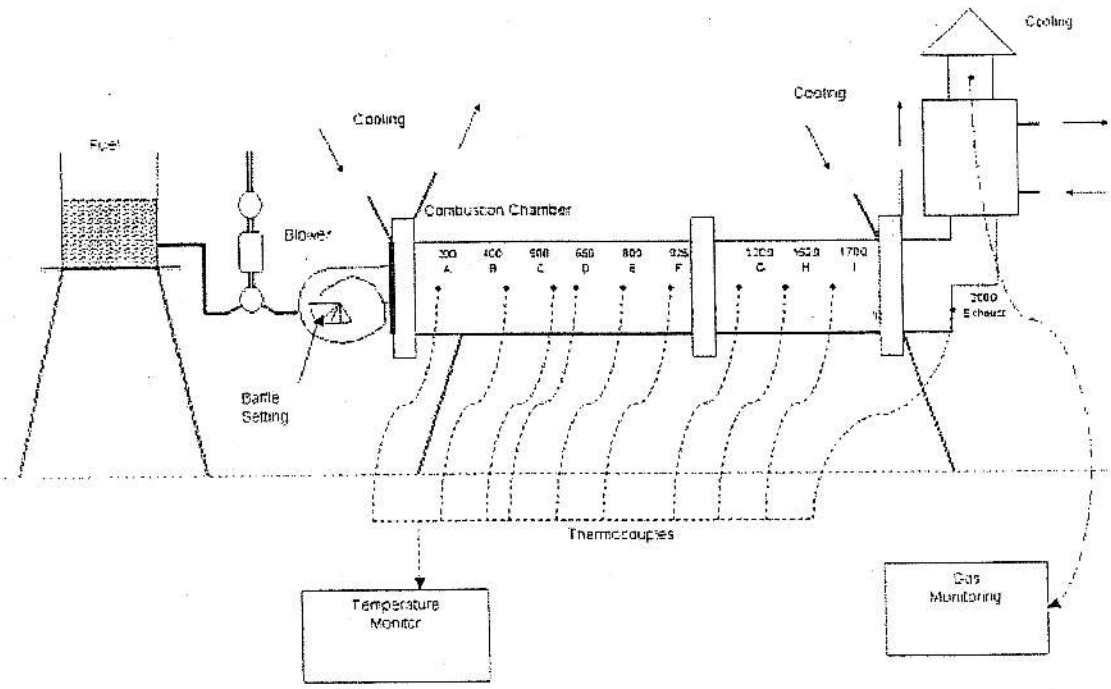
Fatty acid methyl esters (FAME) is being used as alternative fuel, in gas turbines, as they have similar properties to Diesel fuel, FAMEs are alternative fuels capable of being directly used in a turbine and can be blended in various proportions with Diesel fuel (generally 5% or 30% FAME in Diesel fuel). However, low viscosities are needed allow to spray the fuels more easily. Unlike the other fuels, the vegetable oils and the ash pyrolysis oil have a very high viscosity, which can pose a problem to spray the oils in the combustion chamber of gas turbines; however, these fuels may be heated up to decrease their viscosity. **(Iskender et. al 2004)**

The high C/H ratio (high carbon content) of the ash pyrolysis oil from wood, the vegetable oils and the methyl esters may pose a problem of deposits in the combustion chamber of gas turbines. **(Sierra et. al 2005)**

Burning bio-oils in gas turbines is also receiving an increased interest. There are however only few studies on the later subject. **Boucher et al. (2000)** has burned approximately 13 000 liters of bio-oil in a 2.5MW gas turbine pilot unit and concluded that the technical limitations to the process could be overcome by optimizing the bio-oil properties in terms of alkali and ash content, viscosity and heating value and by modifying the gas turbine engine system. The hot section components can be damaged by high-temperature corrosion due to the ash present in the biofuel, which reduces the protective oxide surface film of the alloy. **(Aulisa et. al 2004)**

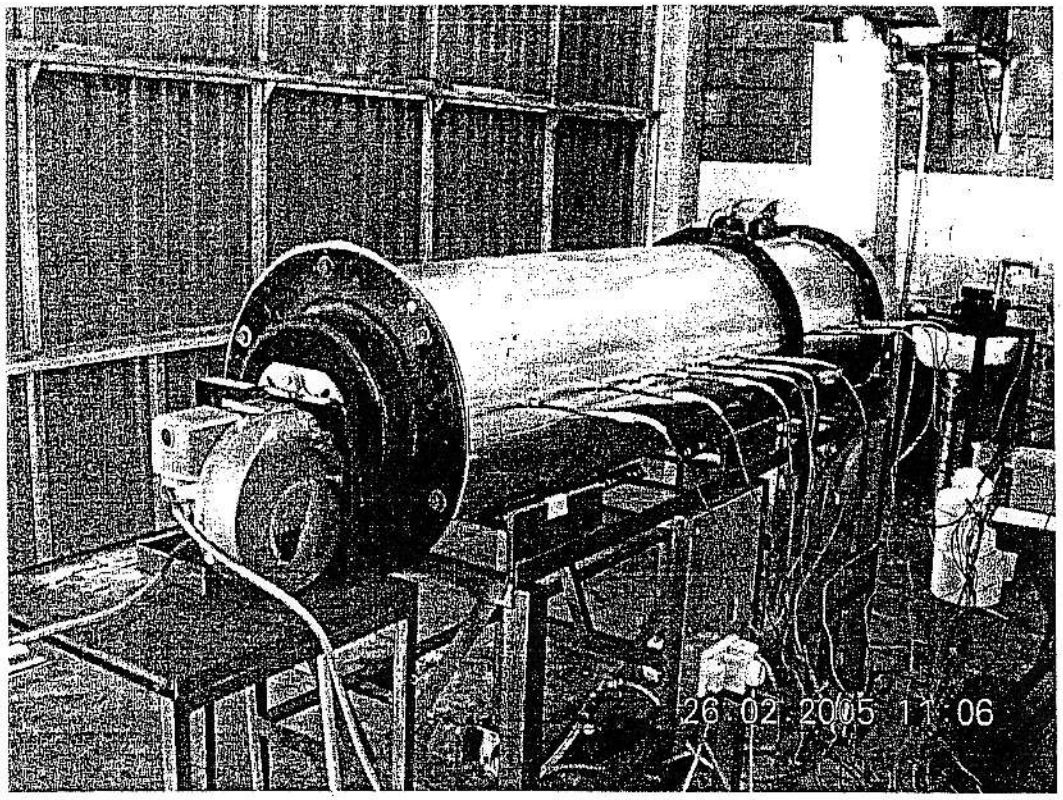
2.0 EXPERIMENTAL TEST RIG

A schematic diagram of the experimental setup is shown in Figure 1 . The experimental setup mainly consists of a burner, combustor and injection system. The setup is being installed in the Combustion Laboratory of Universiti Teknologi Malaysia. A Riello brand 40G Series –G10 light oil industrial burner is being used as a combustion source. The motor of the burner is being used to drive both the air supply fan and fuel pump. The air flow rate was regulated with the help of a damper attached to the inlet section of the fan. The maximum pressure of the fuel pump was limited to 12 bar. The heat output of the burner ranged from 54 to 120 kW, which was subjected to the fuel consumption of the burner. The flame size of the burner could be regulated by adjusting the combustion head setting. The combustion head was connected to a regulating rod, which was extended towards the back of the burner. The fuel reservoir was kept 750 mm below the burner. Before the oil can enter the burner, the oil was filtered and measured using a calibrated float type diesel flow meter.



A circular pilot scale combustion chamber is being used for the experiment. The length and outside diameter of the combustor were 1765 mm and 390 mm respectively. The material was

2.5 mm mild steel sheet. The combustion chamber was being lined with 50 mm refractory material of KIMCAST LW 11. For inserting temperature and emission probes, a series of tapings were made. The actual experimental setup is being shown in Figure 2.



3.0 SCOPE OF WORK

The scope of the work was two fold. Firstly a number of experiments were carried out to evaluate the combustion behavior of distillate oil and other blended fuels with diesel. Using the experimental results, the CFD modeling work was done.

For CFD modeling, a 3D model of the combustor, available in the UTM, was developed so that a reasonable agreement can be reached between the computational and experimental results. This firstly required preparation of 'PREPDF" file based on the elemental analysis of the different fuel compositions used for spray combustion.

To get a converged solution in combustion problem is a long and tedious job as the solution is also dependent of the grid quality and convergence criteria. Therefore, the initial part of the task was a patience job so that a logical solution could be obtained. Later on the problem was progressively solved to achieve a final solution. Initially the problem was solved for 100% diesel only for flow field. Later on the solution was extended to solve the problem for turbulence, species transport and heat transfer which took a long time until the solution was obtained. Similar runs were made or 80D/20P, 60D/40P and 50D/50 PO. The results have been dealt in detail later part of this report.

In order to make the CFD analysis of actual gas turbine, the grid will need to be more fine and the problem size will be big, keeping in view of the size of the actual gas turbine This will in turn results in large computational time and greater convergence problem but it is manageable in some reasonable time.

4.0 MODELLING OF SPRAY COMBUSTION OF LIQUID FUEL

Liquid fuels have always been of interest to gas turbines due to their advantage that liquids are easily storable as compared to gaseous fuels. Disadvantage is that liquid fuel has to be sprayed, vaporized and mixed with air. Combustion occurs at some stage of mixing and ignition. Depending on the efficiency and design of these processes the combustor performs better or worse with a view to emission of nitric oxides, unburnt hydrocarbons or soot.

For the benchmark case, diesel was chosen as a fuel. The input parameters for FLUENT were obtained from the experimental work for the spray combustion which was carried out on the combustion test rig in the Universiti Teknologi Malaysia. Figure 1 shows the design of the combustion test rig.

The model that can suitably define gas turbine or spray combustion is the Premixed Combustion model (PCM). The PCM is used for cases with a single, perfectly premixed reactant stream. In premixed combustion, fuel and oxidizer are mixed prior to ignition. Combustion occurs as a flame front propagating into the unburnt reactants. The overall rate of propagation of the flame is determined by both the laminar flame speed and the turbulent eddies.

The effect of turbulence is to wrinkle and stretch the propagating laminar flame sheet, increasing the sheet area and, in turn, the effective flame speed. The large turbulent eddies tend to wrinkle and corrugate the flame sheet, while the small turbulent eddies, if they are smaller than the laminar flame thickness, may penetrate the flame sheet and modify the laminar flame structure. Reaction takes place in a combustion zone that separates unburnt reactants and burnt combustion products.

The key to the premixed combustion model is the prediction of the turbulent flame speed normal to the surface of the flame. The following influences the turbulent flame speed:

- Laminar flame speed, which is, in turn, determined by the fuel concentration, temperature and molecular diffusion properties, as well as the detailed chemical kinetics.
- Flame front wrinkling and stretching by large eddies, and flame thickening by small eddies

Since industrial low-emission combustors often operate near lean blow off, flame stretching will have a significant effect on the mean turbulent heat release intensity. To take this flame stretching into account, the source term for the progress variable is multiplied by a stretch factor (G). This stretch factor represents the probability that the stretching will not quench the flame; if there is no stretching (G = 1), the probability that the flame will be unquenched is 100%.

5.0 CFD MODELING OF COMBUSTION TEST RIG

Computational fluid dynamics (CFD) tools have gained increasing popularity in the industry and at universities during the last decade. The commercial CFD codes, normally used in the industry, are expected to manage the modeling of chemical reactions within a wide spectrum of reacting systems. Nevertheless, the codes have their limitations to model chemistry with the detailed chemical-kinetic mechanisms relevant to e.g. combustion processes.
Our simulations were aimed to model the turbulent combustion within sprays. The turbulent combustion is being represented by the 'Species Transport' model, which permits to include the effects of finite rate chemistry. The main difficulties of the numerical simulation of reactive flow phenomena concern the physical complexity of the industrial systems, particularly the wide range of space and time scales within which reactions and flow phenomena occur is problematic. The

time scale needed for a given property to be equilibrated varies for the chemical reactions from the fast chemistry (10^{-5}–10^{-14} s) to the slow "frozen" chemistry (10^{-2}–100 s). To overcome the numerical difficulties in complex reacting flows, a thorough understanding of the interactions between mixing of flows and chemical reaction is thus required

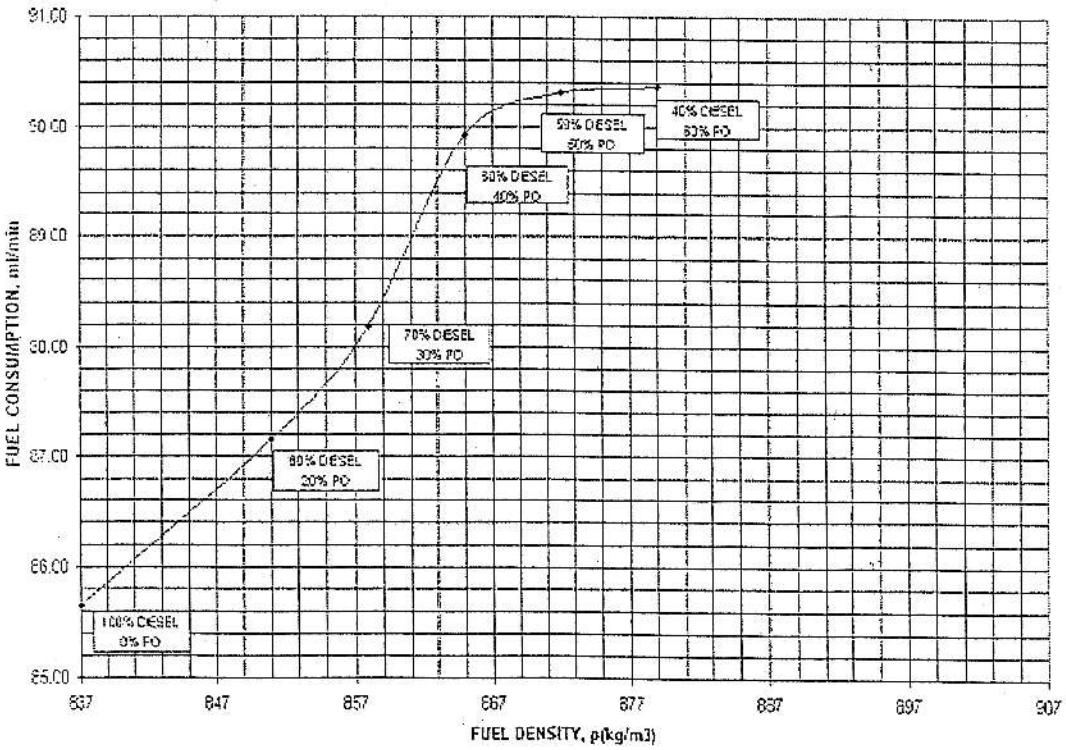
The numerical simulation of heavy fuel oil combustion was a challenging task. In addition to the difficulties inherent to combustion, namely the need to model turbulence, combustion and radiative heat transfer, heavy fuel oil combustion presents other complex phenomena. These include the atomization, spray combustion and soot formation, among others.

We have used FLUENT 6.1 to simulate the reacting flow in the palm olein fired combustion test rig. The standard κ-ε is being used. The CFD-code is being compared against experimental data for the combustor. The combustion data include measurements of gas species concentrations (O_2, CO_2, soot and NOx).

FLUENT has given an insight to the combustion phenomena as far as the results are concerned One can get the diffusion profile inside the combustor. The same is true for temperature distribution, which is useful very useful in identifying the areas of high thermal stresses inside the gas turbine. One can also have an estimation of the various combustion products formed and their distribution inside the gas turbine. FLUENT has capability to estimate NOx and soot that are important as far as performance evaluation is concerned.

6.0 RESULTS AND DISCUSSIONS

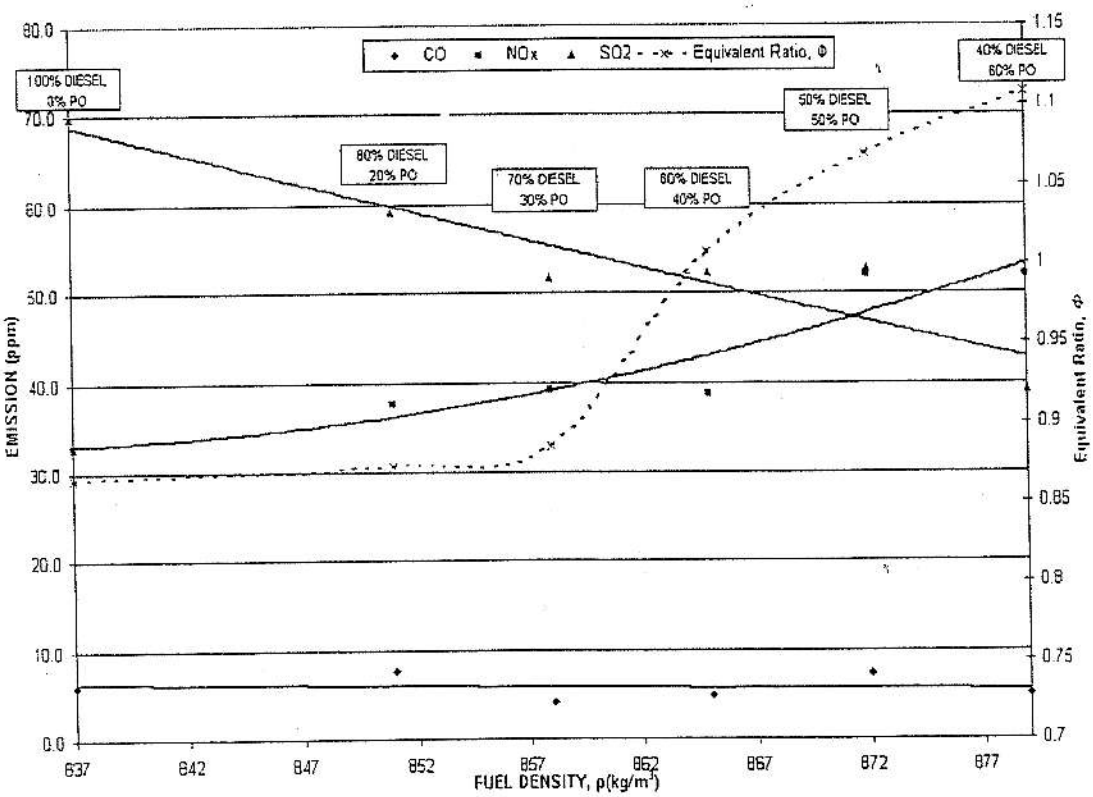
As stated earlier a series of experiments were carried out to estimate the combustion behaviour of liquid fuels. Various blended fuels were made at the Combustion Laboratory at UTM and it was concluded that the fuel consumption increases when the palm oil percentage increases in the blended fuels. The fuel consumption increases to about 6% when the percentage of palm olein increases from 0% to 60% in the diesel fuel. This is being illustrated in Figure 3. This is due to the consumption of more fuels since the calorific values are lower to maintain the desirable loads. The pressure of the fuel pump is set to about 12 bars.

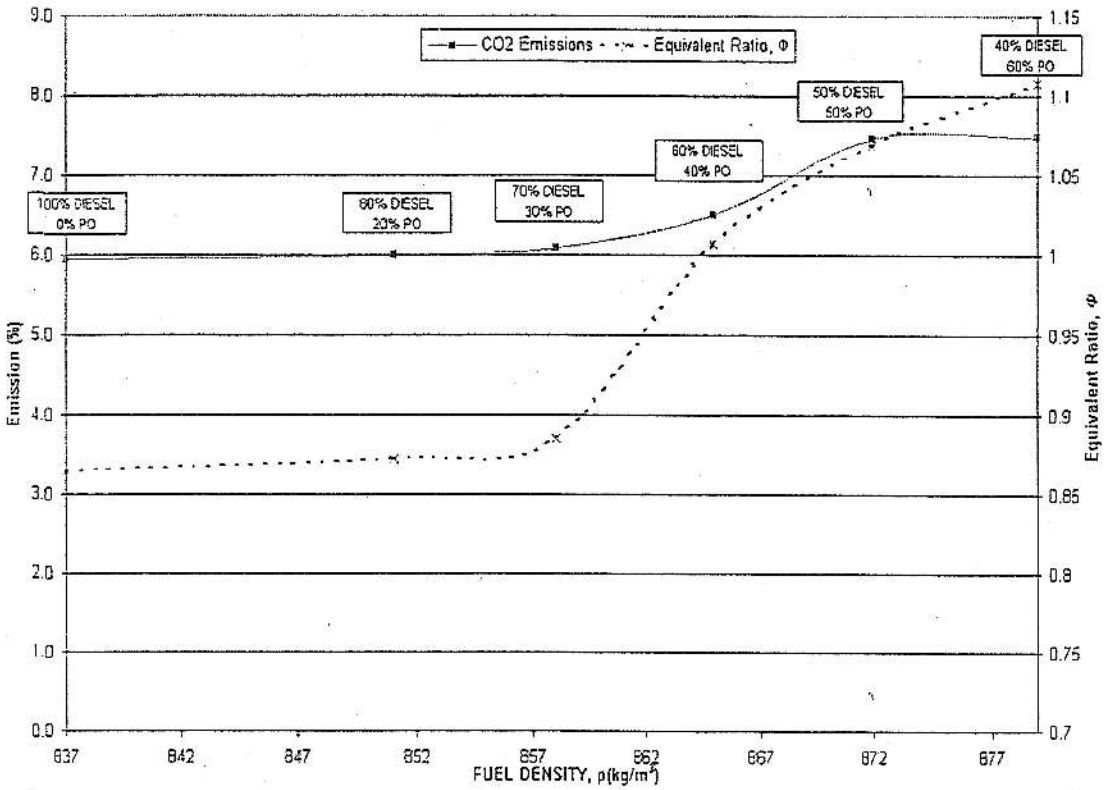


The concentration of exit NOx increases with a decrease in fuel volatility for any given set of values of spray parameters. For 100% diesel, the NOx concentration is around 33 ppm. The NOx concentration for 80%D-20%PO, 70%D-30%PO, 60%D-40%PO, 50%D-50%PO and 40%D-60%PO is 38ppm, 39 ppm, 39 ppm, 52 ppm and 53 ppm respectively. The higher NOx content is because of the increase in peak temperatures of the flame correspond to the increase of equivalent ratios.

The SO_2 emission has a decreasing trend For 100% Diesel, the SO_2 concentration is around 70 ppm at lean condition. The SO_2 concentration decreases as the amount of decreases in the blended fuels. The decrease of Sulphur content in the blended fuel occurs when the palm olein percentage increases from 0% to 60% is the main factor contributing to this decreasing trend.

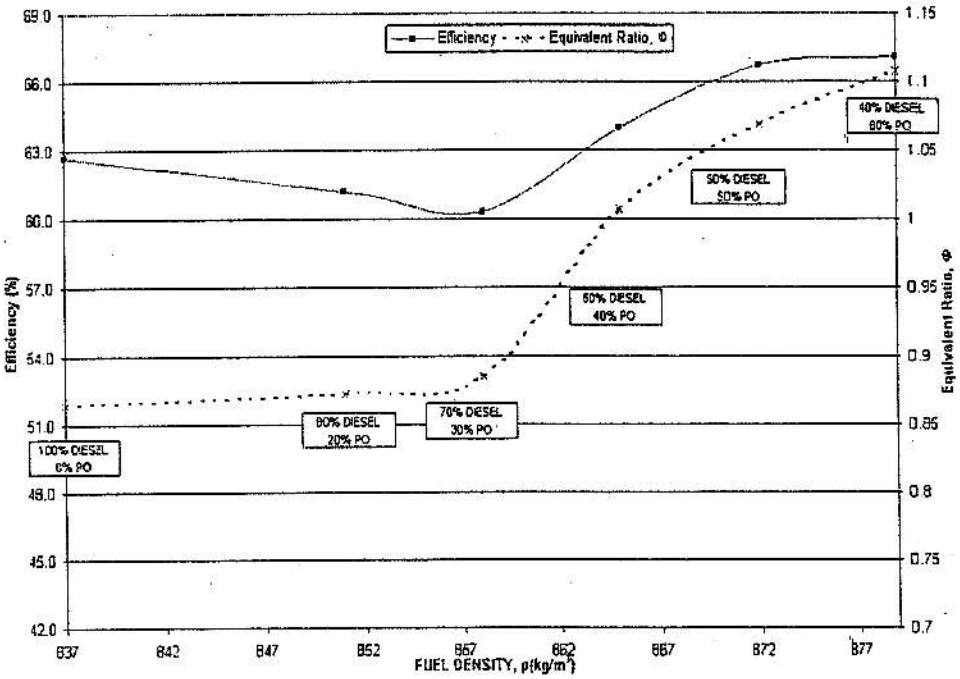
The CO emissions recorded in this series of sets are all below 10 ppm. The graph shows a rather constant trend when the palm olein percentage increases from 0-60% in the blended fuels. The average readings of CO for various blended fuels altered between 4 ppm to 8 ppm. The emission profiles are being illustrated in Figure 4.

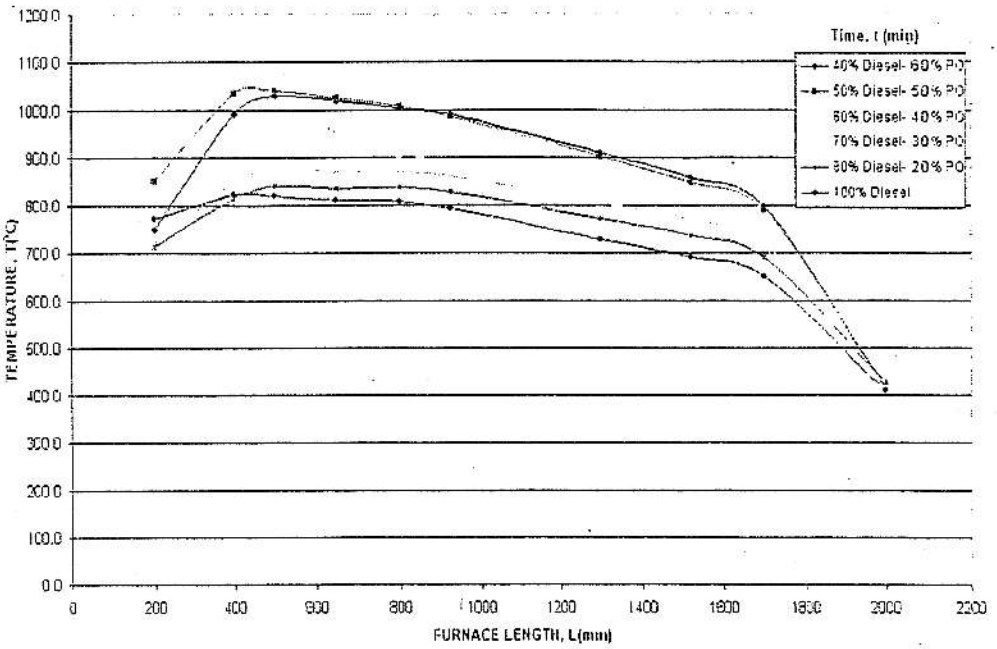




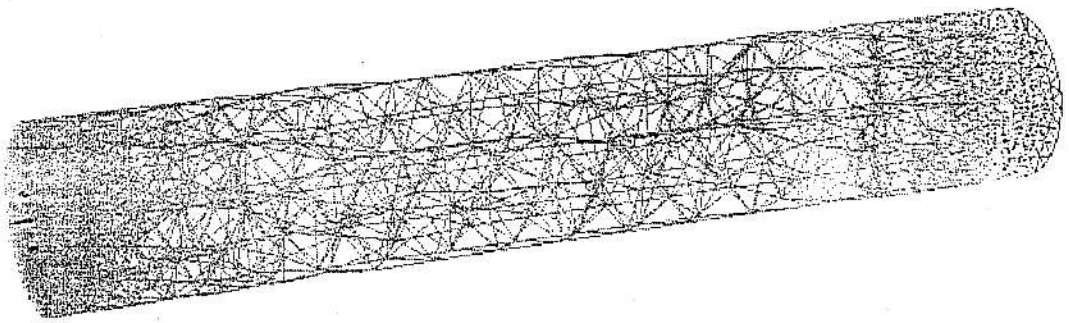
This is being illustrated in Figure 6.

The surface temperature profiles for the diesel and other blended fuels show similar trends. The peak surface temperature near the injection zone increases when the blending of pal olein is increased. For 100% diesel, the peak surface temperature is 822.5 °C while for 80D/20PO, 70D/30PO, 60D/40PO,50D/50PO and 40D/60PO, the peak surface temperatures are 840.0 °C, 875.6 °C, 938.3 °C, 1040.3 °C and 1028.4 °C respectively. The increase of the peak surface temperature is about 25%. The increased equivalent ratio from 0864 (100D) to 1.108 (qt 40D/60PO) is the main reason of increased temperature. This is being illustrated in Figure 7.


Efficiency
Equivalent Ratio, Φ
Efficiency (%)
Equivalent Ratio, Φ
FUEL DENSITY, ρ(kg/m³)
100% DIESEL 0% PO
80% DIESEL 20% PO
70% DIESEL 30% PO
60% DIESEL 40% PO
50% DIESEL 50% PO
40% DIESEL 60% PO
69.0
66.0
63.0
60.0
57.0
54.0
51.0
48.0
45.0
42.0
1.15
1.1
1.05
1
0.95
0.9
0.85
0.8
0.75
0.7
837
842
847
852
857
862
867
872
877



In order to perform the CFD analysis, the geometry o the combustion test rig was drawn in PRO-ENGINEER. The drawing was imported in Gambit and meshing was done. The 3D computational grid consisted of approximately 20,000 computational cells. The meshed grid is being shown in Figure 8.



The conservation equations for the conservation of mass, momentum, fuel mixture fraction were solved for the selected turbulence model. The pressure field was calculated from the continuity equation using the SIMPLE algorithm. However, the convective terms for all the other quantities (velocities, mixture fraction, etc.) were discretized using the second order upwind scheme. For the fuel and air spray three different kinds of configurations were tried. These configurations are being shown in Figure 9.

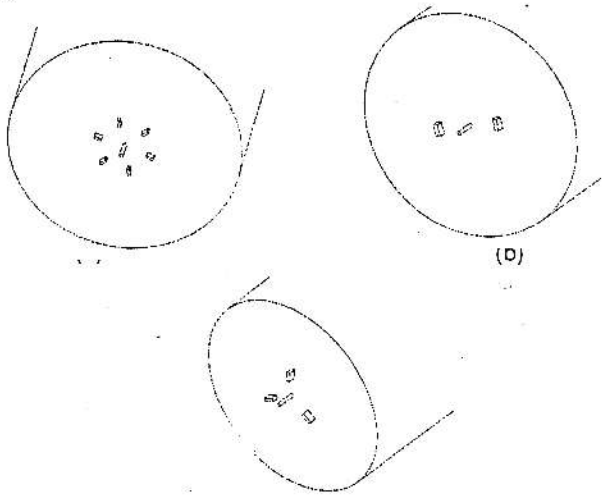


Figure 9. Fuel and oil inlet configurations

The accuracy of the quantitative or even the qualitative trends of the results relating to combustion parameters depends on the accuracy with which the velocity, temperature, and species concentration fields are determined from the numerical computation of the present model.

The flame spreads in the radial direction in the primary zone depending mainly upon the cone angle of the spray. The radial wall jets help to reduce the liner wall temperature appreciably, near the injection points, from where the wall temperature increases gradually in the downstream direction due to diffusion of heat. Due to a lack of information on spray parameters in the reported experimental work, the spray parameters considered for the comparison are chosen after several trials taken over the mean drop diameter and spray cone angle.

According to the temperature contours, heavy fuel oil droplets reach the boiling temperature close to the burner after which fast evaporation follows. The presence of droplets perturbs the flame geometry, the flame front becomes wrinkled and the flame zone is very thick.

Typical temperature contours for diesel and other blended fuels are being shown in Figure 10. The structure of a diffusion spray flame located in the stabilization zone close to the injector can be seen in the temperature contours. If the droplets are small enough they will be completely vaporized before reaching the flame zone. We can see a rich premixed flame starting from the root of the diffusion flame and surrounding the spray of droplets. This is being illustrated in Figure 11.

The temperature distributions predicted by the present model show a fair agreement with the empirical results both at upstream and downstream of the combustor. However, the predicted temperature distribution at the downstream location shows an increasing trend near the wall,

unlike that obtained in the experimental results. This may possibly be attributed to the stick model (according to which the droplets stick to the combustor wall and vaporize there itself) which results in a burning zone near the wall in the downstream part of the combustor.

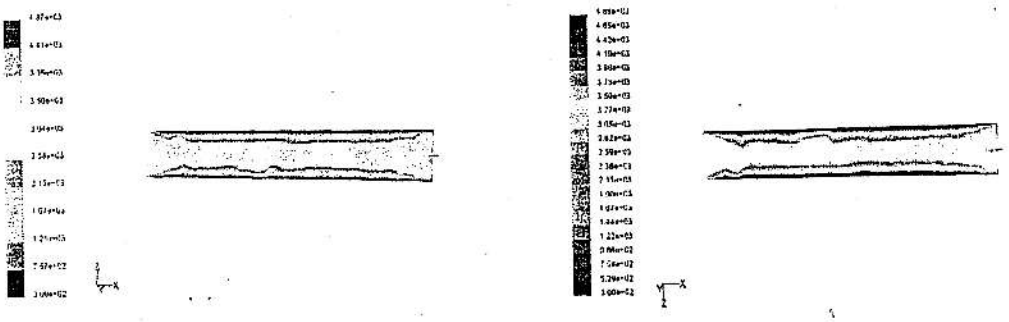
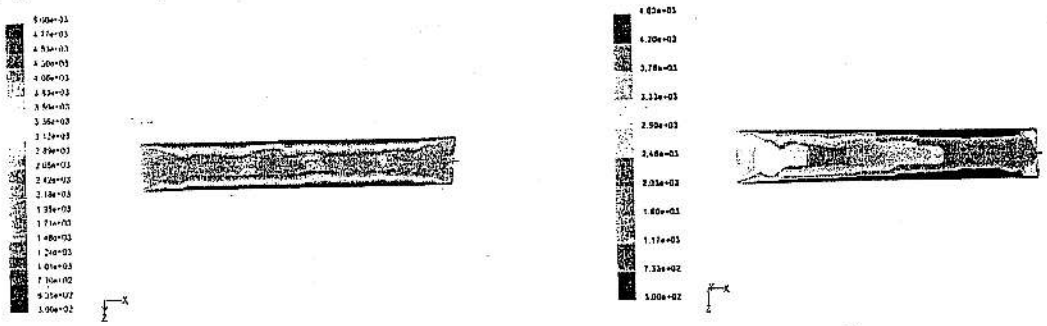


Figure 10. Temperature contours (a) 100% diesel fuel only (b) 80D20PO fuel



(c) (d)

Figure 10 Temperature contours (c) 60D40PO fuel (d) 50D50PO fuel

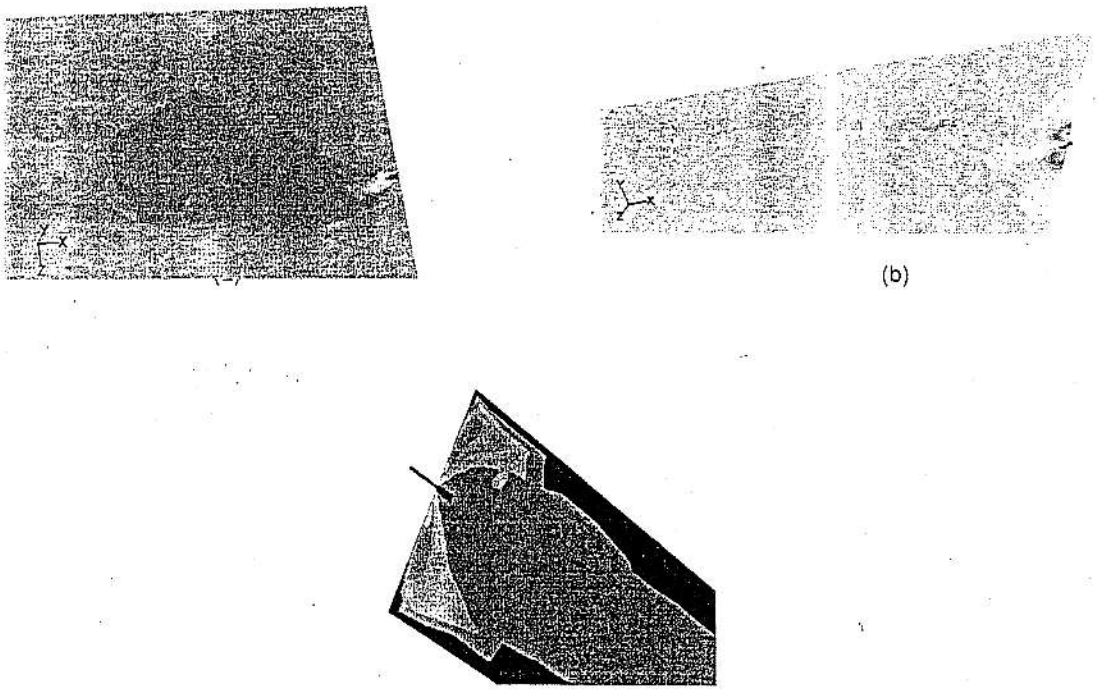


Figure 11. Flame structure for (a) 80D20PO fuel (b) 60D40PO fuel (c) 50D50PO fuel

The success of complete combustion depends on the penetration of fuel droplets, their rate of vaporization, and mixing of fuel vapor with air. For fuel with higher volatility, the combustion efficiency attains a higher value because of a higher rate of fuel vaporization, while the reverse is the case for lower volatile fuels because of poor rate of fuel vaporization. The models available in FLUENT for modeling pollutant formation takes account of the NOx and soot formation.

NOx emission consists of mostly nitric oxide (NO). Less significant are nitrogen oxide (NO_2) and nitrous oxide (N_2O). NOx is a precursor for photochemical smog, contributes to acid rain, and causes ozone depletion. Thus, NOx is a pollutant. The FLUENT NOx model provides the capability to model thermal, prompt, and fuel NOx formation as well as NOx consumption due to reburning in combustion systems. It uses rate models developed at the Department of Fuel and Energy, The University of Leeds, England as well as from the open literature.

To predict NOx emission, FLUENT solves a transport equation for nitric oxide (NO) concentration. With fuel NOx sources, FLUENT solves an additional transport equation for an intermediate species (HCN or NH3). The NOx transport equations are solved based on a given flow field and combustion solution. In other words, NOx is post processed from a combustion simulation. It is thus evident that an accurate combustion solution becomes a prerequisite of NOx prediction. For example, thermal NOx production doubles for every 90 K temperature increase when the flame temperature is about 2200 K. Great care must be exercised to provide accurate thermo physical data and boundary condition inputs for the combustion model. Appropriate turbulence, chemistry,

radiation and other submodels must be applied. To be realistic, one can only expect results to be as accurate as the input data and the selected physical models. Under most circumstances, NOx variation trends can be accurately predicted but the NOx quantity itself cannot be pinpointed. Accurate prediction of NOx parametric trends can cut down on the number of laboratory tests, allow more design variations to be studied, shorten the design cycle, and reduce product development cost. That is truly the power of the FLUENT NOx model and, in fact, the power of CFD in general.

The NOx emissions for diesel and other blended fuel is being tabulated in Table 1.

Table 1: NOx estimations for experimental and simulated conditions

Fuel type	Experimental Value ppm	Simulation result ppm
80D20PO	39	45
60D40PO	39	68
50D50PO	52	122

In laminar flames, and at the molecular level within turbulent flames, the formation of NOx can be attributed to four distinct chemical kinetic processes: thermal NOx formation, prompt NOx formation, fuel NOx formation, and reburning. Thermal NOx is formed by the oxidation of atmospheric nitrogen present in the combustion air. Prompt NOx is produced by high-speed reactions at the flame front, and fuel NOx is produced by oxidation of nitrogen contained in the fuel. The reburning mechanism reduces the total NOx formation by accounting for the reaction of NO with hydrocarbons. The FLUENT NOx model is able to simulate all four of these processes.

The concentration of exit NOx increases with a decrease in fuel volatility. An increase in formation on NO takes place whenever a high temperature of combustion gases prevails which may be due to delayed combustion associated with either a reduction in droplet vaporization rate or increased penetration of vaporizing droplets beyond the primary zone, or an accumulation of fuel vapor in the vicinity of liner wall. Any or all of the above phenomena usually occur for fuels with lower volatility and coarser droplets.

Simulations result shows that the measured and predicted values of NOx are in good agreement for diesel and 80D20PO. The NOx concentration lies between 39–122 ppm. It is evident that k-ε model yields a reasonable prediction of the overall NO emission.

FLUENT provides two empirical models for the prediction of soot formation in combustion systems. In addition, the predicted soot concentration can be included in the prediction of radiation absorption coefficients within the combustion system. You can include the effect of soot on radiation absorption when you use the P-1, discrete ordinates, or discrete transfer radiation model with a variable absorption coefficient.

FLUENT predicts soot concentrations in a combustion system using one of two available models:

- the single-step model , in which FLUENT predicts the rate of soot formation based on a simple empirical rate.
- the two-step model, in which FLUENT predicts the formation of nuclei particles, with soot formation on the nuclei.

In both models, combustion of the soot (and particle nuclei) is assumed to be governed by the combustion rate. Both soot formation models are empirically-based, approximate models of the soot formation process in combustion systems.

Much of the knowledge of the chemistry of soot formation and oxidation comes from measurements made on premixed flames. Premixed flames have relatively rapid gas-phase reaction rates due to their large concentrations of free radicals. In the presence of these fast gas-phase reactions, the rate-limiting steps for soot formation and oxidation typically include the mass transfer processes between solid and gas phases such as particle nucleation, surface growth and oxidation. In such circumstances, the available soot surface area limits the formation and destruction of soot and becomes an integral part of the mechanism controlling soot formation and oxidation.

Soot oxidation is treated by a global, fuel-independent mechanism that is a function of mixture fraction and temperature. Although soot oxidation has traditionally been modeled as a surface area-dependent process, in this work it is modeled as a homogeneous process because diffusion of molecular oxygen is apparently the controlling process.

Table 2 : Soot estimations for experimental and simulated conditions

Fuel type	Simulation result ppm	Percentage Increase
Diesel only	12.5	-
80D20PO	25.5	104
60D40PO	33.3	30.5
50D50PO	77.8	133

The soot formation and oxidation model used in FLUENT contains a number of adjustable constants. Due to this large number of parameters, acceptable agreement between prediction and experiment can usually be achieved by adjusting different combinations of parameters. In general, the usefulness of a model decreases as the number of adjustable constants is increased. However, a model retains its practicality if the constants are global

7.0 CONCLUSIONS

Numerical simulations of liquid fuel spray combustion in a laboratory spray combustor have been done to recognize the influences of inlet spray parameters on important combustion and emission characteristics for fuels with different blending. A reasonable comparison is being made between the experimental and computational results. This will be helpful in future parametric study of the combustor performance under different operating conditions.

A fairly good comparison between numerical simulations and experimental findings were obtained as far as the flame structure is concerned. According to the results of the simulations, it was found that as the palm olein contents increases, fuel oil droplets reach the boiling temperature close to the burner after which fast evaporation follows.

The κ-ε turbulence model is being able to predict the flow field adequately as the selection of turbulence model is important for the simulation of swirl flow. The flow and temperature contours suggests that for fuel with lower palm olein content or high volatility, the combustion process is more uniform because of a higher rate of fuel vaporization.

It was observed that the concentration of exit NOx increased with a decrease in fuel volatility for any given set of values of spray parameters. The overall amount of NOx is predicted rather well with the κ-ε model where the NO molar fraction is around 39 ppm for 100 % diesel. As the blending ratio of palm olein is increased the NOx ppm level also increased and for 2080POD, 4060POD and 5050POD it is found to be 45 ppm, 68 ppm and 122ppm respectively. The high ppm levels are due the slightly higher temperature profiles, which may be due to better mixing and combustion conditions. However, it is evident the ppm level of NOx for 5050POD increased sharply.

Simulation results suggest a similar trend of Soot formation as obtained in NOx formation. However, the soot content are a bit higher for 5050POD which suggest a limit for using blended fuel use in gas turbine combustor.

RECOMMENDATIONS

- As far the temperature distribution and flame structure is concerned, it is recommended that the palm olein blending should not be done in excess of 40%.
- NOx emission increases rapidly as the palm olein concentration increases above 40%.
- The soot formation is important for extended turbine life. Excess soot formation can damage the protective oxide layer of combustion chamber and also the soot particles may cause damage to turbine blades. It is recommended that turbine casing should be opened earlier than scheduled to monitor turbine condition.

REFERENCES

Adouane B., Hoppesteyn P. and Jong W. (2001) Gas turbine combustor for biomass derived LCV gas, a first approach towards fuel-NOx modelling and experimental validation. *Applied Thermal Engineering* Vol. 22: 959-970.

Aulisa E, Barletta A., Gallipoli M., Terenzi A. and Zanchini E. (2004) CFD analysis and overheating control of a turbine International *Journal of Thermal Sciences*. **Article in Press**

Boucher, M., Chaala, A. and Royab C. (2000) Bio-oils obtained by vacuum pyrolysis of softwood bark as a liquid fuel for gas turbines. Part I: Properties of bio-oil and its blends with methanol and a pyrolytic aqueous phase *Biomass and Energy*. Vol. 19: 337-350

Iskender Gokalp, Etienne Lebas (2004) Alternative fuels for industrial gas turbines (AFTUR), Applied Thermal Engineering 24, 1655–1663.

Sharma N.Y. and Som S.K. (2004) Influence of fuel volatility and spray parameters on combustion characteristics and NOx emission in a gas turbine combustor *Applied Thermal Engineering* Vol. 24: 885-903

Sierra, F.Z., Kubiak J., Gonza´lez, G., G. Urquiza (2005) Prediction of temperature front in a gas turbine combustion chamber, Applied Thermal Engineering 25, 1127–1140