Proc. of Int. Conf. on Advances in Robotic, Mechanical Engg. and Design, ARMED

Analysis of Mixture Parameters in a Diesel Engine Combustion Chamber

Padmalochan Prusty
GIET Baniatangi/Mechanical Engineering, Bhubaneswar, India
Email: padmalochan18@gmail.com

***Abstract*—The fuel-air mixture behaviour in a diesel engine was successively analyzed with computational fluid dynamics (CFD) software FLUENT 6.3. The CFD analysis was carried out for a hemi-spherical type combustion chamber cavity with direct fuel injection of diesel fuel and producer gas. The combustion chamber model, with the essential orifices for air and diesel fuel inlets, is tested for a single-cylinder diesel engine of 5.2 kW brake output capacity. The meshing of the combustion chamber was carried out using GAMBIT 2.3. The standard k-ε combustion model was used in the analysis. The CFD analysis examines the changes that occur in the temperature, density, mole fractions of carbon monoxide, carbon dioxide, nitrogen, methane and hydrogen due to composition of diesel and producer gas. The CFD simulations are validated with experimental data obtained from an inbuilt hemispherical shape combustion chamber, and a good agreement between them was noticed. The results show that the modelling has provided a good insight into the flow details and has paved way in optimization in the geometrical design of combustion chamber to get a good diesel-air mixing efficiency, hence the improved engine performance.**

***Index Terms*— Bowl-in-piston, Computational fluid dynamics, Combustion chamber, Fuel-air mixture, Temperature.**

I. INTRODUCTION

A modern compression ignition (CI) engine is expected to meet ecological and economical requirements. It must have low fuel consumptions, low maintenance costs, and also, need to operate under prescribed emission regulations. In a conventional diesel engine, air is admitted into the combustion chamber (CC) and is compressed. Diesel fuel, is then, injected into the CC at extremely high pressure shortly before top dead centre (TDC). This fuel is atomized and heated when get contact with the hot charge in the cylinder. As a result, it rapidly undergoes pre-flame reactions and self-ignites. The process of combustion and further the production of pollutants and noise emissions are mainly controlled by the process of fuel injection. A lot of effort is put into the development of new and improvement of existing diesel fuel injection systems. The injection and the fuel spray characteristics are connected with the combustion chamber geometry which controls the combustion and pollutant formation processes. Therefore, the engine operation characteristics are improved by improving combustion chamber geometry [1].
In engineering, modelling a process means to develop and use the appropriate combination of assumptions and equations that permit critical features of the process to analyze. The modelling of engine processes continues to develop as our basic understanding of the physics and chemistry of the phenomena of interest

Grenze ID: 02.ARMED.2018.7.504

steadily expands and as the capability of computers to solve complex equations continues to increase. Whether a model is ready to pass from one stage to the next depends on the accuracy with which it represents the actual process, the extent to which it has been tested and validated, and the time and effort required to use the model for extensive sets of calculations and to interpret the results [2].

With the continued development of computing power, multidimensional modelling has become an increasingly feasible and economical tool in engine design and development processes [3]. Good computer models are used to achieve a better understanding of the combustion process within engines, finding effective measures to overcome operational problems, evaluating new design concepts, and reducing hardware prototype and development costs. Compared with the extensive research on the modelling of diesel engines, there are only a few references [1, 3–9] on the CFD simulation of diesel engines. This is mainly due to the complexity of the combustion processes while most of these are related to direct injection (DI) diesel engines. Industrial applications of CFD are not restricted to fluid flow and heat transfer only but it can conveniently be used in case of combustion and chemical reactions also.

The objective of the present work is to analyze the fuel-air mixture behaviour in a diesel engine with the inbuilt hemispherical shape CC, and also, with a newly modelled bowl-in-piston CC. Initially experiments were carried out with hemispherical shape CC to analyze and observe the data for CFD analysis. This was followed by CFD analysis using FLUENT 6.3. Finally, the results obtained through these two analyses are compared, and discussed.

II. The Combustion Chamber

The experimental diesel engine test unit available at IIT Guwahati is a single cylinder, four-stroke, water cooled DI diesel engine. The basic engine specifications are given in Table 1. The diesel engine is fitted with hemispherical type combustion chambers (Figure 1). The performance of the diesel engine is examined and observed at different engine loads and speeds.

TABLE I. ENGINE SPECIFICATIONS OF DIESEL ENGINE TEST UNIT

Type	Four cylinder, four stroke, water cooled, direct injection, Bajaj make Tempo, model D-301
Bore	87.5 mm
Stroke	110 mm
Compression ratio	17.5:1
Rated power	5.2 kW@1500 rpm
Loading type	Eddycurrent, Saj make, Pune, model AG-10 with loading device
Air box	With orifice meter and manometer
Fuel tank	15 litres capacity with metering column
Rotameter	For water flow measurement
Temperature indicator	Digital, *K* type temperature sensors (6 points)
Overall size	W 2000 x D 2500 x H 1500 mm
Weight	300 kg (Maximum)
Calorimeter type	Pipe in pipe with rotameter
Computerised	Air flow, fuel flow, temperatures and load measurements
Software package	Performance plots, $P\theta$ and PV plots

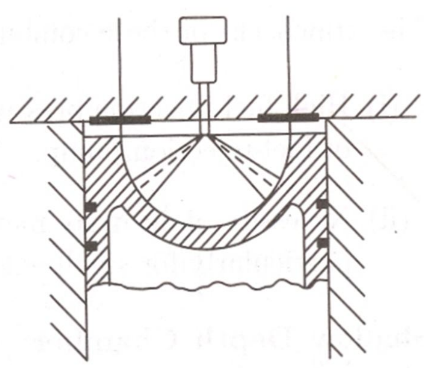

Figure 1 Hemispherical combustion chamber [2]

III. CFD Analysis

A. The Problem

Figure 2 shows the cross-sectional view of a hemi-sphrical type CC. The geometrical modelling is prepared by using Auto-CAD 2007 software. In this type air is inducted through the inlet valve which represents the large diameter. Liquid diesel fuel is injected through the injector of 1.2 mm diameter. The diameter of bowl-in piston cavity is taken as 50 mm. The valve head diameters are assumed as 0.42–0.44 and 0.34–0.37 to cylinder diameter for inlet and exhaust respectively [2].

The meshing of the above CC is done using tetrahedral element (spacing 0.1mm) in GAMBIT 2.3 software. Here, tetrahedral can only be used as parallel faces that are required to mesh with hexagonal elements. After meshing 18833 nodes and 96980 tetrahedral cells are created as shown in Figure 3.

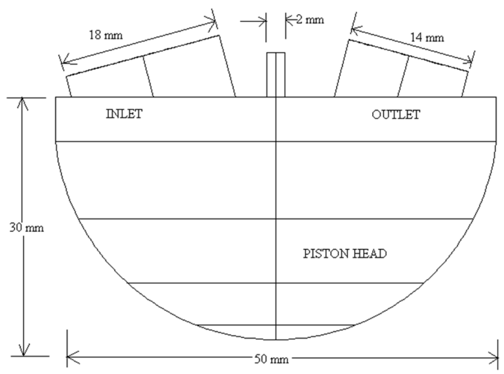

Figure 2 Cross-section of bowl-in piston type combustion chamber using Auto-CAD

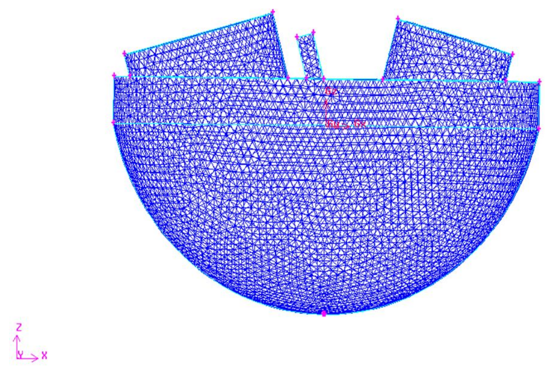

Figure 3 Meshing of bowl-in-piston combustion chamber using GAMBIT

B. The Flow Conditions

The inlet flow condition is considered as steady state flow for the CC simulation. For specifying boundary types, inlet valve and fuel injector are categorized under mass flow inlet type as the mass flow rates of air and fuel are known from the observed experimental diesel data. The outlet valve is specified as outflow type as no information is known about exhaust. The cross-section face is symmetrical about y-axis, and hence, it is kept under symmetry type. All the other faces viz., piston head and CC head are taken as walls. The k–ε approach for turbulence is used for obtaining numerical solution. The standard k-ϵ model is used as it has become the workhorse of practical engineering flow calculations in the recent times [10]. The equations are solved for unsteady incompressible flow [8].

C. The Computational Approach

Turbulence consists of fluctuations in the flow field in time and space. It is a complex process, mainly because it is three dimensional and unsteady. It can have a significant effect on the characteristics of the flow. Turbulence occurs when the inertia forces in the fluid become significant compared to viscous forces, and is characterized by a high Reynolds Number. The k-ε model of turbulence is widely chosen for fluid flow analysis, where 'k' represents the turbulence kinetic energy, and is defined as the variance of the fluctuations in velocity, and 'ε' represents the turbulence eddy dissipation. To simulate the turbulence parameters, a standard k-ε model has been chosen with isothermal heat transfer condition at 300 K. The solver uses k-ε model with two new variables and the continuity equation.

The general transport equation is given by,

$$\frac{\partial(\rho\Phi)}{\partial t} + div(\rho\phi u) = div(\Gamma grad\phi) + S_\phi \qquad (1)$$

Where, Φ = any variable (mass, momentum and energy) per unit mass,
S = Volumetric rate of generation or destruction of species, and
Γ = Diffusion coefficient.

Total mass fractions of all species (fuel and oxidants) equals to unity, i.e.

$$\sum_{all\ species\ j} m_j = 1 \tag{2}$$

Transport equations for k-ϵ model are

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}\left(G_{k+C_{3\varepsilon}}G_b\right)-C_{2\varepsilon\rho}\frac{\varepsilon^2}{k}+S_\varepsilon \tag{4}$$

where, k = turbulence kinetic energy,
ϵ = rate of dissipation,
G_k = Generation of turbulence K.E. due to mean velocity gradients,
G_b = Generation of turbulence K.E. due to buoyancy,
Y_M = Contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate,
σ = Turbulence Prandtl No.,
S = Source terms, and,
$C_1\varepsilon$, $C_2\varepsilon$ and $C_3\varepsilon$ are model constants.

The meshed combustion chamber (Figure 3) is now imported to FLUENT 6.3 using 3D solver. The present problem is solved using non-premixed combustion model as the fuel and air are non-premixed before entering into CC. The discretization of momentum and continuity equations is done by using pressure-based solver, which uses SIMPLE pressure-velocity coupling algorithm. The standard compositions for diesel fuel ($C_{12}H_{23}$) are taken as 87% carbon and 13% hydrogen while for air it is 78% nitrogen and 22% oxygen. And for producer gas compositions are taken as 48.5% nitrogen, 20% carbon monoxide, 18% hydrogen, 12% carbon dioxide and 1.5% methane. All the other properties of fuel and air are auto determined by FLUENT itself. The input temperature for air is taken as atmospheric temperature that is 300 K and that for fuel is taken as 610 K which is the auto-ignition temperature of diesel [11].

IV. CFD RESULTS

Once the properties are given as inputs, FLUENT 6.3 calculates the variations of mean temperature, and mole fraction of carbon monoxide (CO), carbon dioxide (CO_2) and methane (CH_4), with respect to mean diesel-air mixture fraction. Figure 4 shows the variation of mean temperature with mean fuel-air mixture fraction. The maximum mean temperature is observed as 2400 K for diesel from the CFD analysis which is within the theoretical maximum combustion temperature limit of 2900 K. Figure 5 shows the variation of mean density with mean fuel-air mixture fraction. The variations of mole fraction of CO, CO_2, N_2, CH_4 and H_2 with mean mixture fraction are shown in Figure 6–10. The maximum mole fraction of CO is found nearly 0.27 for diesel. The mole fraction of CO concentrations increases with an increase in combustion temperatures [7, 12]. As the temperature increases CO_2 dissociates to form CO and hence mole fractions CO_2 decreases as combustion temperature increases (Figure 7). The variation of mole fraction of CH_4 with diesel-air mixture fraction is shown in Figure 9.

The boundary conditions for the analysis of various contours in the CC are taken from experimental data. The fuel flow rate of 0.0004 kg/s at full throttle and the air flow rate of 0.006 kg/s are considered. The maximum temperature at walls is taken as 2400 K whereas the minimum is ambient temperature. The mixture operating pressure is taken from the observed experimental data as 36.6 bar. Then, the iteration process was started using first order accuracy. The contours of temperature resulted from this analysis are shown in Figures 11–12. Figure 13 shows the variation of total temperature along the computational domain and it is observed that it gradually increases along the domain as we move from inlet valve to exhaust valve.

After simulation the maximum temperature for diesel is found to be 1650 K and that for producer gas is 984 K.

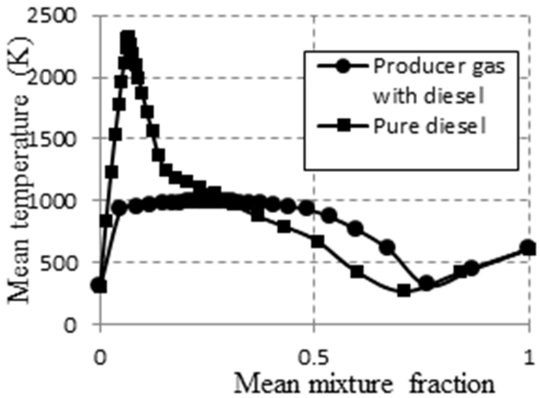

Figure 4 Variation of mean temperature with mean mixture fraction

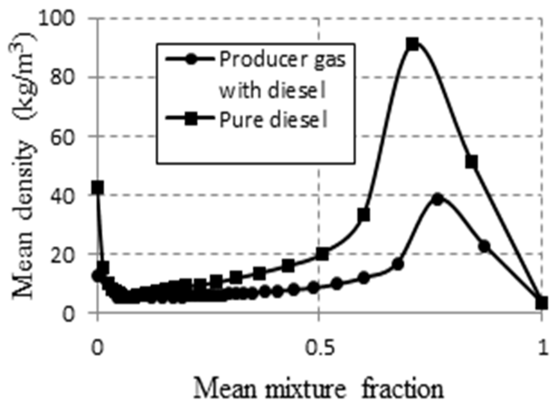

Figure 5 Variation of mean density with mean mixture fraction

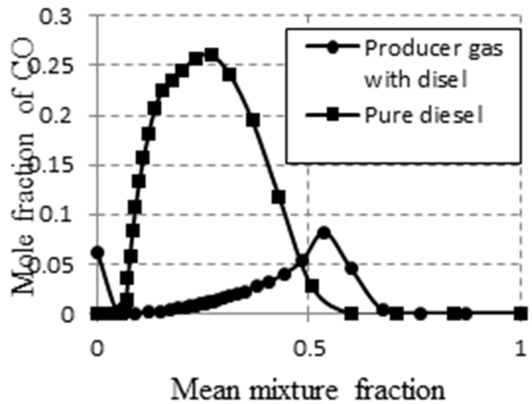

Figure 6 Variation of mole fraction of CO with mean mixture fraction

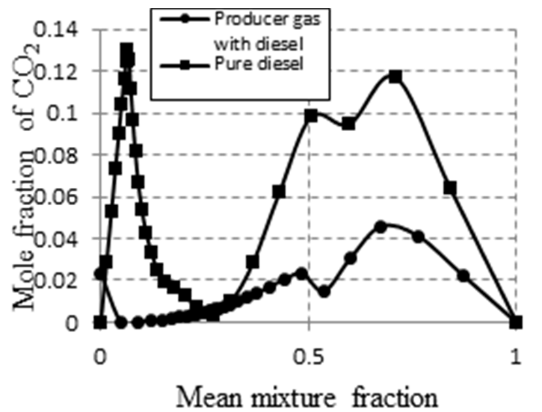

Figure 7 Variation of mole fraction of CO_2 with mean mixture fraction

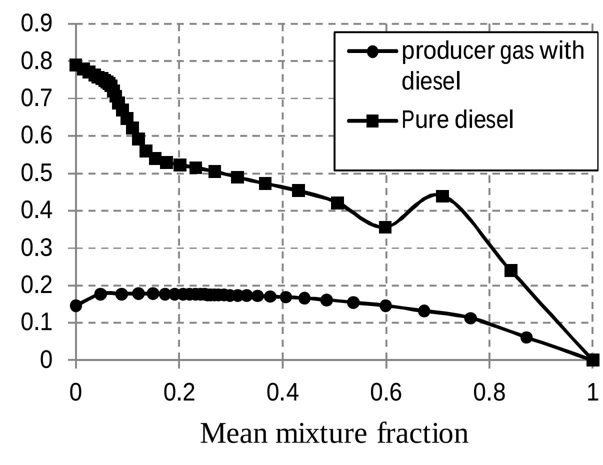

Figure 8 Variation of mole fraction of N_2 with mean mixture fraction

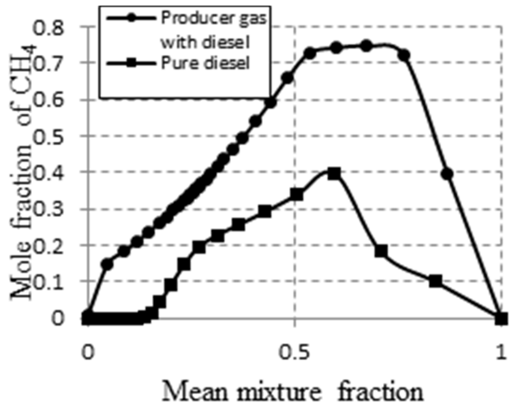

Figure 9 Variation of mole fraction of CH_4 with mean mixture fraction

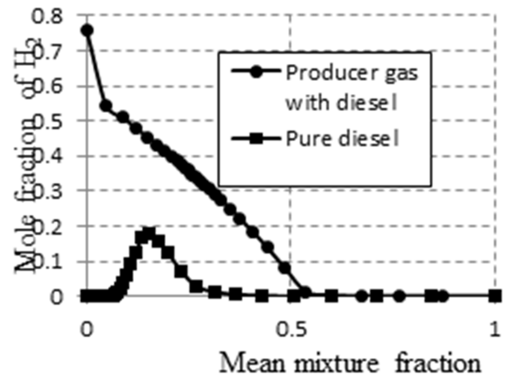

Figure 10 Variation of mole fraction of H_2 with mean mixture fraction

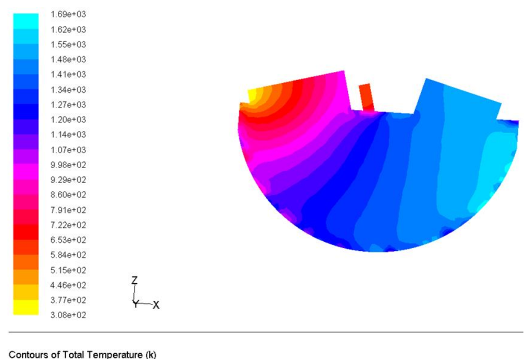

Figure 11 Contours of total Temperature of Symmetry plane for Diesel

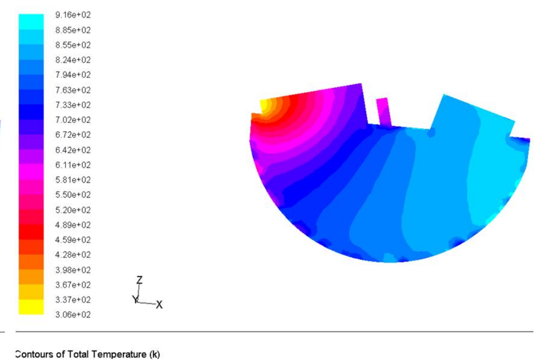

Figure 12 Contours of total Temperature of Symmetry plane for Producer gas

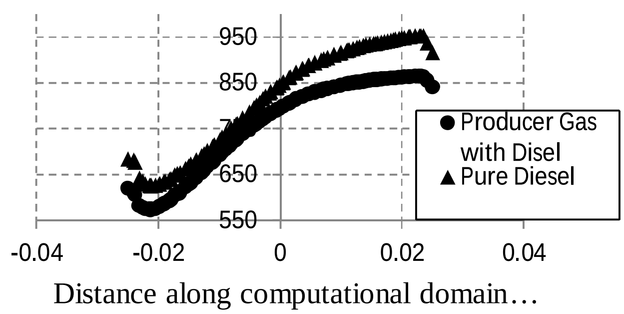

Figure 13 Variation of total temperature along computational domain

V. Conclusions

The present work is aimed at improving the performance of a diesel fuelled DI diesel engine. The computational fluid dynamics is used to examine the changes that occur in the temperature, and the mole fractions of CO, CO_2, and CH_4 due to geometry of combustion chamber. The contours of distributions of energy, temperature, density, and specific heat of diesel-air mixture are also analysed using FLUENT 6.3 software. The results of this work are summarized as follows:

1. The maximum combustion temperature is observed as 2321 K for the hemi-spherical CC. This temperature is validated by comparing with the maximum theoretical combustion temperature limit

of 2900 K for the hemispherical type combustion chamber.

2. The mole fraction of CO concentrations increases with an increase in combustion temperatures. Hence, the concentration of CO_2 deceases with the increase in combustion temperatures.
3. Mole fractions of CO, CO_2 and N_2 are more for diesel than for producer gas.
4. Mol fractions of CH_4 and H_2 are more for producer gas than for diesel.

REFERENCES

[1] M. Volmajer and B. Kegl, "Experimental and numerical analysis of fuel flow in the diesel engine injection nozzle". *Journal of Kones, Combustion Engines*, Vol 8, No 1-2, pp 9–16 (2001).

[2] J.B. Heywood, *Internal Combustion Engine Fundamentals*. McGraw Hill international editions, New York (1988).

[3] Y. He and C. J. Rutland, "Modeling of a Turbocharged DI Diesel Engine Using Artificial Neural Networks," *SAE* Paper No. 2002-01-2772 (2002).

[4] T.R. Anil, S.D. Ravi, M. Shashikanth, P.G. Tewari and N.K.S. Rajan, "CFD Analysis of a Mixture Flow in a Producer Gas Carburetor". *International Conference on Computational Fluid Dynamics, Acoustics, Heat Transfer and Electromagnetics CFEMATCON-06*, July 24-25, Andhra University, INDIA (2006).

[5] S. Jeong, Y. Minemura and S. Obayashi, "Optimization of combustion chamber for diesel engine using kriging model". *Journal of Fluid Science and Technology*, Vol. 1, No 2, pp 138–146 (2006).

[6] M. Masood, M.M. Ishrat and A.S. Reddy, "Computational combustion and emission analysis of hydrogen–diesel blends with experimental verification". *International Journal of Hydrogen Energy*, 32, pp 2539 – 2547 (2007).

[7] M. Masood and M.M. Ishrat, "Computer simulation of hydrogen–diesel dual fuel exhaust gas emissions with experimental verification". *Fuel*, 87, pp 1372–78 (2008).

[8] S.S.Vinay, S.D. Ravi, G. Prema Kumar and N.K.S. Rajan, "Numerical and experimental modelling of producer gas carburettor". *Proceedings of the International Conference on 'Advances in Mechanical Engineering, December 15-17, S.V. National Institute of Technology*, Gujarat, India (2008).

[9] C. Liu and G.A. Karim, "Three-dimensional computational fluid simulation of diesel and dual fuel engine combustion". *Journal of Engineering for Gas Turbines and Power*, 131, 012804 (2009).

[10] B.E. Launder and D.B. Spalding, "*Lectures in Mathematical Models of Turbulence*". Academic Press, London, England (1972).

[11] W.W. Pulkrabek, *Engineering Fundamentals of the Internal Combustion Engine*, Pearson Education, New Delhi (2003).

[12] Z. Sahin, "Experimental and Theoretical Investigation of the Effects of Gasoline Blends on Single-Cylinder Diesel Engine Performance and Exhaust Emissions". *Energy & fuels*, Vol. 22, No 5, pp. 3201-3212 (2008).