Numerical Study of Cofiring Biomass with Coal in Cyclone Combustor.

I. I. Zulkepli1, H. Hasini, A. M. Ikram, M. Z. Yusoff, A. Abd Rahman, A. H. Shamsuddin.

Centre for Renewable Energy, College of Engineering, Universiti Tenaga Nasional, Putrajaya Campus, Km7, Jalan Kajang-Puchong, 43009 Kajang, Selangor, Malaysia.

E-mail: irieyana@uniten.edu.my

Abstract. This paper investigates the flow and temperature distribution inside a cyclone combustor during combustion of coal and coal blend. The combustion under study is based on the actual cyclone combustor experiment rig used to test the performance of combustion when it be different coal-biomass blend is used. Experiment investigation on 100% coal combustion was also carried out and serves as a basis for the subsequent test for co-firing of different coal or biomass blend. Validation of temperature magnitude along cyclone furnace at 100% coal combustion condition shows good agreement between the measured and CFD results. Subsequent simulation of coal and biomass blend shows very good impact as it gave less error compared to experiment.

Keywords: Computer Fluid Dynamic (CFD); Empty Fruit Bunch (EFB); cofiring.

1. Introduction

Global warming is one of the serious consequences of the Greenhouse gas (GHG) emissions which could be attributed to the released of carbon based element into the atmosphere has become global concern since many decades ago. Coal is known to be an important source of and its reserve is estimated to be greater than the other type of fossil fuels [1]. Thus many countries around the world are shifting their fuel utilization by utilizing coal as part of the energy security measure. The benefit of coal is to ignite the combustion inside the combustor and it can be consider as largest reserve and cheapest option at this moment [2]. Coal contributes approximately 40% worldwide electricity generation to be main important role in the energy supply [3]. However, since the coal technologies are available and widespread, demand of the coal increased in the global market and the amount of it become lapse gradually. For the purpose to cover up the coal utilization, the co-firing term were introduced, it’s actually mixture coal with biomass to achieve approximately 100% coal at the power plant. Using biomass as few portions to replace pulverised coal represent the greatest potential in existing coal fired power plant. Cost-effective can be save rather than use 100% biomass in the small boiler. Palm residues are abundant in Malaysia due to the fact that Malaysian palm based industry is very large. The palm waste could be divided into three sections which is Empty Fruit Bunch (EFB), Palm Kernel Shell (PKS) and Mesocarp Fiber [4]. Palm residue is suitable as the main biomass element because of the high and comparable heating value.

2. Governing equation

CFD solve the Reynolds Average Navier-Stokes equation in addition to the continuity and energy equations as shown in (1).

\[
U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ V \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \bar{w} U_i \right]
\]

The turbulence is resolved using the k-epsilon (k-\( \varepsilon \)) turbulent model which consist of turbulence kinetic

---

1 To whom any correspondence should be addressed.
energy and dissipation energy equation ($\varepsilon$) are given by (2) and (3) [5]. $k$-$\varepsilon$ standard involve turbulence kinetic energy ($k$) and ($\varepsilon$) symbolize to dissipation energy equation that used for stress-strain

Relationship [6] [7]

$$\overline{u_i u_j} = \frac{2}{3} k \delta_{ij} - V_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$

(2)

Calculate from

$$V_i = \frac{k^2}{\varepsilon} \text{ and } C_\varepsilon = 0.09$$

(3)

$V_i$ is represent transport equation which is solution by $k$ and $\varepsilon$. $C\mu$ instead of constant in (4,5 and 6).

$$u_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{V_i^2}{\varepsilon} \right) \frac{\partial k}{\partial x_j} \right] + P_k - \varepsilon$$

(4)

$$u_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \nu + \frac{V_i^2}{\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{\varepsilon 1} \frac{k}{\varepsilon} P_k - C_{\varepsilon 2} \varepsilon$$

(5)

With, $P_k = -\overline{u_i u_j} \frac{\partial U_i}{\partial x_j}$

(6)

The coefficients values are,

$$\sigma_k = 1.00, \quad \sigma_\varepsilon = 1.30, \quad \sigma_{\varepsilon 1} = 1.44, \quad \sigma_{\varepsilon 2} = 1.92.$$

In the numerical setup, the near wall function setup as non-equilibrium and assumes inflexible temperature. Non-premixed combustion includes all the data from the fuels into the inlet at the combustor, and it known as pre-probability density function (Pre-Pdf). Pre-PDF is to import both fuels into combustion process and it has been modelled using non-adiabatic process [5]. A discrete phase model (DPM) used to track trajectories of particles as shown in (7),

$$\frac{du_i}{dt} = F_0(u_i - u_i^T) + g_i(p_i - \rho) / \rho_p + F_i / \rho_p$$

(7)

3. Model Description

3.1. Cyclone Combustor

The cyclone type combustor is designed with several tangential inlets, located at the primary and secondary chamber [8]. It is able to produce highly turbulence and mixing of air and fuel as injected into the furnace. The mixture environment inside it also determines the firing shape to minimize corrosion on the furnace wall [9]. For the first stage at the primary chamber, fuel has devolatilization process but it will be complete combust in the secondary chamber. Figure 1 is top view of cyclone combustor and the flow inside it. The model developed at Gambit (CFD pre-processor software) before run into Fluent (CFD solver). The full shape and geometry followed the original cyclone combustor from Cardiff University and go through certain modification to be suit with fuel type in Malaysia.

Figure 1. Top view of Cyclone Combustor.  
Figure 2. Model Cyclone Combustor in Gambit 3D
3.2 Mesh
Using Hexagonal type of the mesh structure on the geometry can assist it to generate fast and accurate [10]. Geometry selection have very acute angles for each volume and this condition must used Tetrahedral mesh which is proper for 3D boundary mesh and unstructured geometry containing only triangular faces [10].

3.3 Boundary condition and Model assumption.
For primary chamber temperature is required 800K-1000K lower than the secondary temperature. It is of the combustion started, with devolatization of the fuel than it continues burn completely in the secondary chamber. Both chambers have inlets with different velocity because it includes air that has been divided by 25% and 75% individually with direction normal to boundary. Behavior on the combustion flow and temperature distribution inside the chambers, the model of solver must be noted using pressure based, 3D spaces, absolute velocity formation. Model that has been develop effects the turbulence swirling flow and temperature distribution at every sector inside the combustor. Modeling design is used steady system and activates the energy to see the results at the end of the simulation for the combustion process. Pre-PDF imported in this process to run the simulation until the temperature react and compared with experiment data. The error of the comparison must be lower than 10% for validation which is acceptable for combustion. Error is following the researcher in combustion field.

4. Result and Discussion.
Result at the end of the simulation shows the temperature data has been satisfied with the error percentage below 10% which is acceptable for combustion. The graph in figure 3 shows simulation data compared to the experiment and the model successfully working. The temperature is taken at the secondary chamber is using the thermocouples that installed.

![Figure 3. Experiment value versus simulation value of temperature distribution.](image)

![Figure 4. Behaviour flow of the 100% coal combustion.](image)

Simulation on CFD on figure 4 is according to the ultimate analysis of Adaro coal, as shown in table 1. The data followed the ASTM standard and comparison based on the Cardiff University experiment on 100% Adaro coal [11]. Swirling flow of the combustion arrangement makes the bottom of the secondary chamber gain high temperature. It is because complete combustion happens at the region and require more heat to burn all the fuels. Top of it shows low temperature as reason fuels that burn become ash and release to the cyclone exhaust. Continues work has been done by the simulation of combination 97.5% Adaro coal and 2.5% EFB as shown in figure 5 and 95% Adaro coal and 5% EFB as shown in figure 6 which is using the existing model of 100 % Adaro coal as validation. The ultimate analysis for cofiring is based on the calculation of the ultimate analysis data for both fuels. The cooling zones appear at the bottom as the secondary chamber because relative air enters to make the fuel burn rapidly. Highest temperature for figure 5 and figure 6 is above 1600°C it’s considered acceptable as combustion with the biomass fuels normally unstable. The contour shows the reactions
between coal and biomass as in red in colour (high temperature) appear at the few sections at the bottom secondary combustor and become cool at the top.

**Table 1.** Ultimate Analysis Adaro Coal

| Ultimate Analysis | %   |
|-------------------|-----|
| C                 | 67.05 |
| H                 | 6.44  |
| N                 | 0.9   |
| S                 | 0.12  |
| O₂               | 20.41 |
| Gross CV (MJ/kg)  | 24.65 |

5. Conclusion

The combustion behaviour in cyclone combustor move the fuels and air together in swirling flow and give high temperature at the bottom of the secondary chamber. The fuels go through complete combustion at the bottom of the secondary chamber and leave the ash at the top of it. Model of the 100% Adaro coal successfully working and can be validation to the cofiring process as it give 6.2% error to the experiment result which is acceptable for the combustion process. Standard (k-ε) turbulence model assist the simulation concept similar with the experimental result.

Acknowledgement

Authors would like to acknowledge for the financial supports given: (1) Long Term Research Grant (01201102 LRGS) from Ministry of Higher Education (MOHE) and Universiti Sains Malaysia (USM)

References

[1] Hashimoto N, Kurose R, Tsuji H, Shirai H 2007 Energy & fuels 21(4) 1950-1958
[2] P.D.O. Energy Energy Resources 22 July 2012 [Online] http://www.doe.gov.ph/ER/Coal.htm. [Accessed 13 November 2012]
[3] Pallarés J, Gil A, Cortés C, Herce C 2009 Fuel Processing Technology 90(10) 1207-1213
[4] Shamsuddin H 1989 RERIC International Energy Journal 11(2) 25-33
[5] Ikram A, Mostafa N A, Rahman A, Shamsuddin A H 2010 The 6th International Conference on Combustion Incineration Pyrolysis and Emission Control Kuala Lumpur
[6] Bakke A 2002 Lecture 10 Turbulence Model, Applied Computational Fluid Dynamic [Online] http://www.bakker.org/dartmouth06/engs150/10-rans.pdf [Accessed 10 October 2012]
[7] Xia J L, Yadigaroglu G, Liu Y S, Schmidli J, Smith B L 1998 International journal of heat and mass transfer 41(11) 1485-1497
[8] Syred C, Fick W, Griffiths A J, Syred N 2004 Fuel 83(17) 2381-2392
[9] Ikram A, Rahman A A, Mostafa N A, Shamsuddin A H 2009 In Energy and Environment, 2009 ICEE 2009 3rd International Conference on 482-484
[10] Bakker A Lecture 7 Meshing, Applied Computational Fluid Dynamic 2002 [Online] http://www.bakker.org/dartmouth06/engs150/07-mesh.pdf. [Accessed 2012 October 10]
[11] Rahman A 2006 Studies of Co-firing of Coal with Biomass on a Two Stage Simulator for Utility Boilers Cardiff University