CFD-based Analysis of Non-Premixed Combustion Model in Biomass Grate Furnaces

M Hafiz¹, L O Nelwan¹ and M Yulianto¹

¹ Department of Mechanical and Biosystem Engineering, Bogor Agricultural University, Indonesia

E-mail: mohamadhafiz.eng@gmail.com

Abstract. Biomass grate furnace is widely used as heat source for various uses including grain drying. In this study, a CFD simulation using Fluent 18.0 academic was performed on a biomass ladder grate furnace, which can be used later to improve the design as well as the operation technique of the furnace. A downscaled overfeed type furnace with size of 15 x 30 x 50 cm was built to validate the model. The turbulence model used in this study was k-epsilon while the combustion model of non-premixed combustion was used. The simulation was performed with the biomass feed rate of 4 kg/h and air flow velocity of 7.5 m/s at 3.81 cm inlet diameter. The simulation result at outlet temperature was 673 °C and inside temperature were 775 and 717 °C, while the composition of gases was 0.18 for CO, 0.2 for CO₂, 0.001 for CH₄, 0.09 for H₂O, 0.51 for N₂ and 0.029 for other gases. Test results from a biomass ladder grate furnace were used to validate the model and the results are 646 °C for the outlet temperature, 712 and 582 °C for inside temperature. Comparison between simulation and measurement results shows good value with average percentage of deviation 12.12%.

1. Introduction

Biomass grate furnace is one of simple technologies for heating that is frequently used by small scale industries. One way to develop a detailed design of furnace with is by using CFD simulation. Furthermore the simulation can increase the understanding of the processes occurring in the biomass furnace systems. The virtual view inside the furnace delivers a comprehensive overview of the system behavior and can help to solve combustion and operation problems. [1]. Wang and Yan [2] summarizes in an article about researches in CFD for biomass thermochemical conversion and conclude the CFD can predict accurately biomass conversion from thermochemical aspect. CFD can analyze distribution of product, flow, temperature, ash, until emission such as NOₓ. even though it still need model approaching and assumption and resulting a little error.

Some authors have proposed efficient furnace with various biomass fuels. Rene et al [3] developed furnace for boiler with capacity 176 kW with fuel from corn and wheat straw in round bales shape with feeder type batch. Gomez et al [4] developed mathematic model for wood boiler stove with underfeeding system and Nesiadis et al. [5] optimized small scale boiler with two combustion chamber using logwood as fuel. Combustion simulation by fossil fuels has widely used with non-premixed or mixture fraction approaches but it rarely used in biomass combustion appliances so far. Prieler et al. [6] and Mayr et al. [7] showed the high accuracy of the non-premixed in case of methane-air and oxy-fuel combustion. They predicted temperatures, heat flux and fractions of species of combustion in a very good way. Albrecht et al. [8] and Venturini et al. [9] presented biomass combustion in grate furnaces with mixture...
fraction models. Both used the GRI 3.0 methane combustion mechanism for chemistry calculation. Without tars represented in the studies. The mixture fraction approach unites the benefits of the classical species transport models. The advantage of this modelling approach is that complex chemical reaction mechanisms can be implemented at low computational effort.

The main objectives of the present work were to evaluate furnace performance with CFD by non-premixed combustion model with turbulence model and make scenarios for new design to optimize furnace performance.

2. Model description

The present paper using non-premixed combustion as species model with non-adiabatic energy treatment. Fuel and oxidizer enter combustion chamber as separated streams. Mixing and burning take place simultaneously. The reaction chemistry in the Probability Density Function (PDF) method for solving turbulent chemistry interaction is determined using mixture fraction model. The equilibrium model is used and it assumes that the chemistry is rapid enough for chemical equilibrium to always exist at the molecular level [10]. Basing on the simplifying assumptions, the instantaneous thermochemical state of the fluid is related to the mixture fraction f. Species mole fraction is computed based using minimization of Gibbs free energy concept. The mixture fraction f is defined in terms of the atomic mass fraction as Equation 1.

\[ f = \frac{m_i - m_{i,ox}}{m_{i, fuel} - m_{i, ox}} \]  

Where \( m_i \) is the mass fraction for element i. The subscript ox denotes the value at the oxidizer stream inlet and fuel the value at fuel stream inlet. Knowing f we can also calculate mass fraction of fuel and oxidizer at any place x,y,z.

Radiation model P1 used in this research because the model is quite steady to predict temperature and gas composition [11]. For turbulence model, three k-\( \varepsilon \) standard, k-\( \varepsilon \) renormalization group (RNG) and k-\( \varepsilon \) realizable. The major differences in the models are as follows, the method of calculating turbulent viscosity, the turbulent Prandtl numbers governing the turbulent diffusion of k and \( \varepsilon \) and the generation and destruction terms in the \( \varepsilon \) equation [10]. The k-\( \varepsilon \) model for k-\( \varepsilon \) standard, k-\( \varepsilon \) renormalization group (RNG) and k-\( \varepsilon \) realizable are expressed in equation 2, 3, and 4.

\[ \frac{\partial}{\partial x_i} \rho u_i \varepsilon = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \rho k \frac{\varepsilon}{k} - C_{2\varepsilon} \rho k \frac{\varepsilon^2}{k} \]  

(2)

\[ \frac{\partial}{\partial x_i} \rho u_i \varepsilon = \frac{\partial}{\partial x_i} \left[ \alpha_k \mu_{eff} \frac{\partial \varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \rho k \frac{\varepsilon}{k} - C_{2\varepsilon} \rho k \frac{\varepsilon^2}{k} \]  

(3)

\[ \frac{\partial}{\partial x_i} \rho u_i \varepsilon = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + \rho C_{1\varepsilon} \frac{\varepsilon^2}{k+\varepsilon} + C_{1\varepsilon} \frac{\varepsilon}{k} - C_{3\varepsilon} \]  

(4)

In the derivation of the k-\( \varepsilon \) model, k-\( \varepsilon \) standard assumption is that the flow is fully turbulent, and the effects of molecular viscosity are negligible. The standard k-\( \varepsilon \) model is therefore valid only for fully turbulent flows. While the standard k-\( \varepsilon \) model is a high-Reynolds number model, the RNG theory provides an analytically-derived differential formula for effective viscosity that accounts for low-Reynolds number effects. K-\( \varepsilon \) realizable is a modified transport equation for the dissipation rate, \( \varepsilon \), has been derived from an exact equation for the transport of the mean-square vorticity fluctuation [12], [13].

Solution control method was used to choose interpolation scheme in mesh nodes. The pressure-velocity coupling scheme used is the SIMPLE scheme. In this scheme selected Least Squares Cell Based for gradient, Pressure standard and First-Order Upwind Scheme for Momentum, Turbulent Kinetic Energy, Turbulent Dissipation Rate, Energy, Mean Mixture Fraction and Mixture Fraction Variance. Where the First-Order Upwind Scheme is the lightest and fastest converging interpolation scheme.
3. Methodology

Material used in the research was chopped corncob as fuel, furnace build from concrete (brick, cement and sand) and steel for feeder, ladder and outlet. Equipment for record temperature data was Thermocouple Type-K and recorder, and air flow data used anemometer. The main research was simulation for the furnace with non-premixed model with steady state condition until convergent, valid with test performance and then make scenario for better furnace.

3.1. Furnace configuration

The size of the furnace was 30 cm (l) x 15 cm (w) x 50 cm (h) with diameter of screw feeder 7.62 cm, inlet air diameter 3.81 cm and outlet diameter 9 cm. Wall of the combustion and air chamber made by concrete and the combustion ladder, screw feeder, outlet and inlet cylinder made by steel. The schematic diagram of the performance test is presented in figure 1. Chopped corncob used in the test have average diameter and length 0.04 m and 3 cm. The combustion begins by burning corncob on the combustion ladder until the flame steady, and then the corncob insert continuously by the feeder with capacity 4 kg/h. The blower used to flow the fresh air with velocity ± 7.5 m/s or equal to mass flow ± 36.73 kg/h. Thermocouple position for outlet was in the middle of outlet cylinder, about 10 cm (x-axis) and 4.5 cm (y-axis), for combustion chamber the position is 10 cm and 20 cm (x-axis) from outlet wall with 10 cm (y-axis).

3.2. CFD Modelling Configuration

The CFD software used for simulation was ANSYS Fluent R18.0 Student version. Mesh 3D model was done with mesh program from ANSYS itself with curvature advance size function, medium relevance center, medium smoothing, fast transition and medium span angle center. Nodes for the simulation were 33697 nodes with 153988 elements. With boundary condition velocity inlet for inlet air, pressure outlet for gas outlet, mass flow inlet for fuel inlet and wall for furnace wall. Furnace model used in this simulation as same as the furnace for test performance. The position of the fuel inlet is different from performance test like at figure 2, this is because the combustion happened in ladder of the furnace, not in the feeder.

![Figure 1. Schematic view of the performance test used in this study](image1)

![Figure 2. Schematic view of the computational study used in this study](image2)

Fuel used during the experiments was corncob. Table 1 presents properties data (proximate and ultimate) of the fuel used; the value was input in coal calculator facility. Wall used in the simulation are concrete and steel. The density of the concrete and steel used are 1300 kg/m³ and 8030 kg/m³ with specific heat of 750 J/kg.K and 502.48 J/kg.K and thermal conductivity of 1.7 W/m.K and 16.27 W/m.K.
Table 1. Corn cob composition and properties [14]

| Parameter          | Value | Unit    |
|--------------------|-------|---------|
| Proximate analysis |       |         |
| Moisture           | 7.96  | w.b.wt% |
| Ash                | 1.49  | d.b.wt% |
| Volatiles          | 85.08 | d.b.wt% |
| Fixed Carbon       | 13.43 | d.b.wt% |
| Ultimate analysis  |       |         |
| Carbon (C)         | 48.1  | d.b.wt% |
| Hydrogen (H)       | 6.02  | d.b.wt% |
| Nitrogen (N)       | 0.36  | d.b.wt% |
| Oxygen (O)         | 43.96 | d.b.wt% |
| Sulphur (S)        | 0.05  | d.b.wt% |
| Calorific Value    | 18.4  | MJ/kg   |

Thermal conditions of the wall boundary are convection. Heat transfer coefficient for concrete was 15.7 W/m².K, free stream temperature as same as environment temperature 31 °C, internal emissivity 1 and wall thickness 0.12 m, Heat transfer coefficient for steel 10.46 W/m².K, and wall thickness 0.003 m.

3.3. Model Validation

Validation was conducted by comparing temperature of the experimental and simulation data. Accuracy of experimental and simulation result expressed as average percentage of deviation. Value of average percentage of deviation defined as Equation 5.

\[
\text{Average percentage of deviation} = \left| \frac{T_{\text{simulation}} - T_{\text{experimental}}}{T_{\text{experimental}}} \right| \times 100\% 
\]  

4. Results and discussion

The combustion chamber temperature of the test can be seen at figure 3. The test was conducted for 1 hour and 30 minutes, the temperature of the furnace hard to steady because of the problem of the furnace design, where the flame come out from the feeder so the biomass in feeder is combusted and stuck in the middle of the screw feeder. Steady flame was observed in the last 20 minutes, with average outlet temperature of 646 °C, the middle furnace temperature was 581 °C and 711 °C. average inlet temperature of air flow was 39 °C with environment temperature of 31 °C. The inlet air temperature was higher than ambient temperature because the position near feeder with lack of flame so the heat is sucked by the blower.

Figure 4 until figure 7 shows contours and vectors of the velocity. Figure 4 show primary air injection vectors that pass through the chamber, figure 5 and figure 6 show the vector of velocity in outlet furnace and inside the screw feeder. In the screw feeder the air flow entered to the feeding biomass area.
Figure 3. Temperature of furnace test performance

Figure 4. Velocity vectors of primary air

Figure 5. Velocity vectors of primary outlet

Figure 6. Velocity vectors of feeder

Figure 7. Contours of char in the furnace
In the simulation, the biomass was assumed to come out through the ladder. The biomass was injected from the floor by discrete phase model. Figure 7 shows carbon solid ($c_s$) or char of the biomass in the bed region. The char was fly in the furnace and not burned out all. Some of char was come through the feeder. Figure 8 shows contour of the temperature in which the flame rising from the ladder.

![Figure 8. Detail contours of temperature in combustion chamber](image)

**Table 2. Deviation and accuracy between simulation and experimental**

| Turbulent model | Position | Simulation ($^\circ$C) | Experimental ($^\circ$C) | Deviation (%) | Accuracy (%) | Average Accuracy (%) |
|-----------------|----------|------------------------|--------------------------|--------------|--------------|----------------------|
| $k$-$\varepsilon$ Standard | Outlet | 642 | 646 | 0.61 | 99.39 | |
| | Back | 747 | 712 | 5.02 | 94.98 | 92.25 |
| | Front | 684 | 582 | 17.62 | 82.38 | |
| | Outlet | 746 | 646 | 15.38 | 84.62 | |
| $k$-$\varepsilon$ RNG | Back | 982 | 712 | 37.99 | 62.01 | 72.27 |
| | Front | 755 | 582 | 29.82 | 70.18 | |
| | Outlet | 674 | 646 | 4.35 | 95.65 | |
| $k$-$\varepsilon$ Realizable | Back | 783 | 712 | 10.07 | 89.93 | 87.26 |
| | Front | 720 | 582 | 23.79 | 76.21 | |

Table 2 shows the comparison of experimental result vs model. These data represent the time averaged values once steady state had been achieved. The experimental values were collected for approximately 20 minutes from 17:07 – 17:27 at experimental time. From the three models of turbulence used it can be seen the accuracy of the outlet and rear of the furnace is very good for $k$-$\varepsilon$ standard and $k$-$\varepsilon$ realizable, above 90%, but for $k$-$\varepsilon$ RNG only 84% at the output and 62% in part back. The front for the standard $k$-$\varepsilon$ is 82%, 70% for $k$-$\varepsilon$ RNG and 76% for $k$-$\varepsilon$ realizable. To illustrate the visualization of the temperature ratio obtained between the test and the three models of turbulence can be seen in figure 12. From the data it can be seen that the $k$-$\varepsilon$ RNG turbulence model is not suitable for this furnace case, since this model is more suitable in the case of turbulence with the number Reynold is low, and the best turbulence model for combustion is a standard $k$-$\varepsilon$ with an average accuracy of 92.25%, of that value then the simulation is acceptable, so the boundary conditions used in this model will be used in the next furnace design.
Figure 9. Contours of gas O\(_2\)

Figure 10. Contours of gas CO

Gas contour in the simulation was varied, with domination of N\(_2\) for 51%, the O\(_2\) had zero value after pass through the ladder, shown in figure 9 and composition of other was 18% for CO, 20% for CO\(_2\), 0.1% for CH\(_4\), 8% for H\(_2\)O, and 2.9% for other gases. Gas CO\(_2\) and CO emission in the simulation quite high, especially for gas CO shows in figure 10, it still not confirmed for the validation in the test performance but it shown the gas still not combusted completely.

There was one issue in the furnace that can disturb the combustion stability of the fuel in the combustion chamber there was the flame come out from the feeder. When the flame came out from feeder, the fuel pyrolysis in feeder and made the corn cob sticky and stuck in pitch of feeder. A new design of the furnace is a must and in this study, one alternative design of the furnace is evaluated. The alternative design made change the position of the feeder to upper side, add one secondary air inlet and insert partition between the combustion chamber and feeding chamber. The combustion ladder is smaller in the 2\(^{nd}\) furnace with just four ladders. Figure 12 shows flow of air in the furnace, the primary inlet air same was 7.4 m/s and secondary air 1.5 m/s, average outlet flow of the furnace was 6.2 m/s and flow air in the feeder 1.2 m/s.

Figure 11. Contours of temperature in the 2\(^{nd}\) furnace

Figure 12. Velocity vectors of air flow in the 2\(^{nd}\) furnace
Figure 13. Contours of O\textsubscript{2} in the 2\textsuperscript{nd} furnace

Figure 14. Contours of CO in the 2\textsuperscript{nd} furnace

New design of the furnace solved issue of the furnace and help the biomass did not pyrolysis or burned in the feeder. Figure 11 show the temperature in the feeding chamber just around 95 °C and temperature in the feeder still around 75 °C. the temperature enough to prevent the corncob as fuel to be pyrolysis or burned. Average outlet temperature of the new design was 700 °C, with zero concentration of O\textsubscript{2} in the combustion chamber (figure 13) and concentration of gas CO (figure 14) quite high but smaller than original design with composition 0.15.

Based on comparison between original design (1\textsuperscript{st} design) and improved design (2\textsuperscript{nd} design) we can see different value of temperature produced both, the outlet temperature produced by 2\textsuperscript{nd} design higher ± 25 °C with higher gas outlet rate due to additional air, the sensible enthalpy value for both fuels is almost similar and the sensible gas outlet enthalpy is higher in the 2\textsuperscript{nd} design because the outlet temperature is higher so that the resulting heat is higher 2192 J/s or about 21%. Detailed comparison between the two furnaces can be seen in table 3.

Table 3. Comparison of sensible enthalpy and heat of furnace simulation

| Furnace design            | 1\textsuperscript{st} design | 2\textsuperscript{nd} design |
|---------------------------|------------------------------|------------------------------|
| Biomass mass flow (kg s\textsuperscript{-1}) | 0.00113                      | 0.00113                      |
| Biomass sensible enthalpy (J kg\textsuperscript{-1}) | 102494                      | 102500                      |
| Gas flow (m s\textsuperscript{-1}) | 5.2                          | 6.2                          |
| Gas outlet mass flow (kg s\textsuperscript{-1}) | 0.0099                      | 0.0118                      |
| Gas sensible enthalpy (J kg\textsuperscript{-1}) | 824491                      | 876848                      |
| Heat of furnace (J s\textsuperscript{-1}) | 8063                        | 10254                        |

5. Conclusions
This research shown a good CFD simulation of combustion with non-premixed combustion model for corncob fuel, the average accuracy of the model is 92.25% using k-ε standard. Simulation process of furnace can analyze gas composition of the combustion (CO\textsubscript{2}, CO\textsubscript{2}, O\textsubscript{2}, etc.) and velocity of air in the combustion. A new design of the furnace has been designing in this research with higher heat of stove and solve issue of the flame come out from the feeder area.

Acknowledgement
This research is funded by Indonesia Endowment Fund for Education (LPDP) year 2017.
References
[1] M Buchmayr, J Gruber, M Hargassner and C Hochenauer 2016 A computationally inexpensive CFD approach for small-scale biomass burners equipped with enhanced air staging Energy Convers. Manag. 115: 32–42.
[2] Y Wang and L Yan 2008 CFD Studies on Biomass Thermochemical Conversion Int. J. Mol. Sci. 9(6): 1108–30.
[3] R Morissette, P Savoie and J Villeneuve 2013 Corn Stover and Wheat Straw Combustion in a 176-kW Boiler Adapted for Round Bales Energies 6(11): 5760–74.
[4] M A Gómez, J Porteiro, D Patiño and J L Míguez 2015 Eulerian CFD modelling for biomass combustion. Transient simulation of an underfeed pellet boiler Energy Convers. Manag. 101: 666–680.
[5] N Athanasios, N Nikolaos, M Nikolaos, G Panagiotis and E Kakaras 2015 Optimization of a log wood boiler through CFD simulation methods Fuel Process. Technol. 137: 75–92.
[6] R Prieler, M Demuth, D Spoljaric and C Hochenauer 2015 Numerical investigation of the steady flamelet approach under different combustion environments Fuel 140: 731–743.
[7] B Mayr, R. Prieler, M Demuth, M Potesser and C Hochenauer 2015 CFD and experimental analysis of a 115 kW natural gas fired lab-scale furnace under oxy-fuel and air-fuel conditions Fuel 159: 864–875.
[8] B A Albrecht, S Zahirovic, R J M Bastiaans, J A van Oijen and L P H de Goey 2008 A premixed flamelet-PDF model for biomass combustion in a grate furnace Energy and Fuels 22(3): 1570–80.
[9] P. Venturini, D Borello, C Iossa, D Lentini and F. Rispoli 2010 Modeling of multiphase combustion and deposit formation in a biomass-fed furnace Energy 35 (7): 3008–21.
[10] Ansys ANSYS Fluent Theory Guide 2010
[11] X Lu and T Wang 2012 Effect of Radiation Models On Coal Gasification Simulation ASME 2012 Int. Mech. Eng. Congr. Expo 6: 1175–92
[12] T H Shih, W. W. Liou, A Shabbir, Z Yang and J Zhu 1995 A new k-e eddy viscosity model for high reynolds number turbulent flows. Comput. Fluids 24 (3):227–238
[13] R Prieler, M Demuth, D Spoljaric and C Hochenauer 2014 Evaluation of a steady flamelet approach for use in oxy-fuel combustion Fuel 118: 55–68
[14] S Melin 2013 Considerations for Grading Agricultural Residue