Computational Fluid Dynamics Simulation of Horizontal Biomass Gasifier

V. K. Aharwal 1, R.N. Singh 2*

1School of Energy and Environmental Studies, Devi Ahilya Vishwavidyalaya, Indore, India
2School of Energy and Environmental Studies, Devi Ahilya Vishwavidyalaya, Indore, India

*Corresponding Author: rnsingh.seema@gmail.com

Abstract—Computational Fluid Dynamics (CFD) simulation of horizontal biomass gasification process has been carried out. The gas-solid interaction, thermal-flow behavior and biomass gasification process inside a horizontal gasifier were studied using the software named as commercial CFD solver ANSYS CFX. The influence of gasification air velocity, temperature inside the reactor, species mass fractions of biomass, turbulence eddy dissipation and eddy viscosity on performance of horizontal gasifier were examined. All phases are described using an Eulerian approach to model the exchange of mass, energy and momentum. The analysis includes two cases, the first case deals with flow pattern and the influence of geometry whereas the second case deals with reaction modeling in a steady-state thermal analysis. It is found that superficial gas and mass fraction of biomass has a strong influence on the outlet temperature and gas velocity. Producer gas temperature and H2 and CO distributions in producer gas indicate that reactions in the instantaneous gasification model occur very fast and finish very quickly.

Keywords—Aspen plus CFD CFX, turbulence kinetic energy, turbulence eddy dissipation, biomass, horizontal gasifier, kinetic reaction.

I. INTRODUCTION

Modern world and structure of our society are inextricably related to energy production. At present scenario, the global population has become highly dependent on the production of energy through the industrial burning of fossil fuels. Scarcity of fossil fuels has led towards the use of alternative energy sources like solar, wind, hydro power, geothermal and biomass for sustainable development [1]. Biomass is a renewable organic matter (such as agricultural crops, wood and wood waste, organic components of municipal and industrial wastes, animal waste etc.) which has been utilized for energy production for many years. Biomass is an attractive source of energy. Although numbers of routes are available for conversion of biomass into energy, however thermal conversion (through gasification route) of biomass was found more prominent [2, 3, 4].

The most of gasifier available in the markets are vertical gasifier. There designs are easy to control and few moving parts help to operate gasification process smoothly. However these designs have some problems [1]. To overcome on vertical gasifier design, Singh et al 2016 designed and developed batch type 10 kWth horizontal fixed bed updraft gasifier. They have optimized the gasifier design and operational parameters such as length to diameter ratio (L/D) of reactor, zones temperature, fuel consumption rate (FCR), gas production rate experimentally. They have also compared the performance of vertical biomass gasifier (Updraft) with horizontal gasifier [5].

In this paper theoretical approach using mathematical model was used to simulate the effects of the parameters that influence the process of gasification. Pyrolysis zone of gasifier is mathematically described through a coupled of chemical equations such as chemical kinetics, heat transfer and mass transfer. Aspen Plus simulation tool is used for theoretical simulation of the gasification processes of Horizontal gasifier [20].

Mathematical models have certain advantages when compared to experimental method. ANSYS CFD can produce a large number of data points with less experimental data (ANSYS 2009). An ANSYS CFD CFX model can develop accurate geometry of the gasifier reactor [6,7,8,9]. The main purpose of the numerical mathematical models involve understanding and quantifying the thermo-chemical processes during biomass gasification and investigate the influence of the main input parameter such as moisture content, air/fuel ratio, fuel consumption rate, producer gas...
composition, eddy viscosity and zone temperature profile etc. [10,11,12].

CFD model results are capable of predicting qualitative information and in many cases accurate quantitative information. It is a very powerful tool for the design and development of new ideas and technologies.

Generally fluid (gas and liquid) flows are governed by partial differential equations (PDE) which represent conservation laws for the mass, momentum and energy. The basic principle behind CFD modeling is that the simulated flow region is divided into small cells. Differential equations of mass, momentum and energy balance are discretized and represented in terms of the variables at any predetermined position within or at the center of cell. These equations are solved iteratively until the solution reaches the approximately to laminar conditions. CFD modeling techniques are becoming widespread in the biomass thermo chemical conversion areas specifically in biomass gasification and combustion.

II. MODEL DESCRIPTION

Standard k-epsilon Model
The standard k-ε model was employed to simulate the turbulent flow, due to its suitability for a wide range of wall-bounded and free-shear flows. The standard k-ε model is the simplest of turbulence two-equation model in which the solution of two separate transport equation allows the turbulent velocity and length scales, which are to be independently determined. The k-ε model is a semi-empirical model with several constants which could be obtained from experiments.

The standard k-ε model is based on transport equations model for the turbulence kinetic energy (k) and its dissipation rate (ε). The turbulence kinetic energy (k) and its rate of dissipation (ε) was obtained from the following transport equations:

\[
\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho u_j k) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_k + S_k
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho u_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_t}{k} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + C_{2\varepsilon} \frac{\varepsilon^2}{k} + G_\varepsilon - C_{\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon
\]

\[G_k = -\rho u_i u_j \frac{\partial u_i}{\partial x_j}\]

Chemical Reaction Model
Chemical reaction models were used in the CFD simulation for homogeneous gas to gas reactions. The homogeneous gas reaction assumes the carbon species gasified instantaneously, and the carbon is treated as a gas. The inter phase exchange rates of mass, momentum and energy are assumed to be infinitely fast. Carbon particles are made to gasify instantaneously, thus the solid-gas reaction process can be modelled as homogeneous combustion reactions. This approach is based on the locally-homogeneous flow (LHF) model proposed by Faeth, 1987 [13] which, implying infinitely-fast inter phase transport rates. The instantaneous gasification model can effectively reveal the overall combustion process and results without dealing with the details of the otherwise complicated heterogeneous particle surface reactions, heat transfer, species transport, and particle tracking in turbulent reacting flow [14]. The eddy-dissipation model is used to model the chemical reactions. The eddy-dissipation model assumes the chemical reactions are faster than the turbulence eddy transport, so the reaction rate is controlled by the flow motions. The global instantaneous gasification mechanism is modelled to involve the following gaseous species: C, O₂, N₂, CO, CO₂, H₂O, and H₂. All of the species are assumed to mix in the molecular level. In this approach, carbon is modelled as a gas species.

The combustion takes place at temperature range of 800°C to 1200°C. However, heterogeneous reaction takes place between oxygen present in the air and solid carbonized fuel producing carbon dioxide i.e. C + O₂ → CO₂ ........ (3) [15, 16, 17].

Hydrogen in fuel reacts with oxygen in the air and blasts producing steam, which can be expressed as

\[\text{H}_2 + \frac{1}{2} \text{O}_2 \rightarrow \text{H}_2\text{O} \quad \text{(4)}\]

In the gasification, a number of high temperature chemical reactions take place in the presence of limited supply of oxygen. The major reactions in this zone are water gas reaction, the water shift reaction, the boudouard reaction and methanation reaction. The fuel in this process is highly carbonized form and converted to red hot with driven off all the volatile matters. Temperature in the gasification process varied between 600°C and 800°C. Reactions which take place during gasification process could be summarized as follows [18].

Water gas reaction
\[\text{C} + \text{H}_2\text{O} \rightarrow \text{CO} + \text{H}_2 \quad \text{(R₁)}\]

Water shift reaction
\[\text{CO} + \text{H}_2\text{O} \rightarrow \text{CO}_2 + \text{H}_2 \quad \text{(R₂)}\]
Boudouard reaction
\[ \text{C} + \text{CO}_2 \rightarrow 2\text{CO} \]  \hspace{1cm} (R_3)

Methanation reaction
\[ \text{C} + 2\text{H}_2 \rightarrow \text{CH}_4 \]  \hspace{1cm} (R_4)

The rate of heat transfer has a direct relation with eddy diffusivity. Higher the diffusivity value higher the rate of heat transfer through turbulence component and lower the value of diffusivity \[19\]. Mathematically could be written as
\[
\frac{Q}{A} = -\rho C_p \frac{dT}{dy} = -\rho C_p \frac{\partial T}{\partial y} = -\rho C_p \frac{\partial^2 y}{\partial y} \]  \hspace{1cm} \text{(5)}

Where, \( \epsilon_d = \frac{\partial^2 y}{\partial y} \)

### III. Simulation of Horizontal Gasifier

Simulation modeling of horizontal gasifier was done in Creo software which is parametric sketch based software developed by PTC for optimization of the gasifier parameters \[20, 21\]. The CAD model for horizontal gasifier reactor is modeled for two lengths and single diameter. One dimension was taken as length 1200 mm and dia. 275 mm (Fig. 1), however in second model, dia. (275 mm) was kept constant (Table 1) and length was taken as 1000 mm (Fig. 2). In both cases gasifier material was selected as mild steel (MS) with 5 mm thickness for simulation.

#### Table 1: Geometrical description of horizontal gasifier

<table>
<thead>
<tr>
<th>Experiment cases</th>
<th>Outer diameter (mm)</th>
<th>Inner Diameter (mm)</th>
<th>Length (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>275</td>
<td>265</td>
<td>1200</td>
</tr>
<tr>
<td>Case 2</td>
<td>275</td>
<td>265</td>
<td>1000</td>
</tr>
</tbody>
</table>

The model is meshed with tetrahedral elements with relevance set to fine sizing (Fig. 3). Inflation and growth rate set to normal. The elements of geometry near inlet pipe juncture, outlet pipe juncture and biomass inlet pipe was refined for smooth transitions (Fig.4). Total number of elements is 59970 and nodes are 12022 were calculated.

#### IV. Assumptions

Following assumption has been considered
- The analysis was performed in pseudo static stage due to non-availability of mathematical equation for batch process with respect to biomass consumption.
- The analysis was performed under steady state conditions.
- CFD simulation, vertically oriented up-draft gasifier parameters for gasification process was considered.

#### V. Results and Discussion
The whole concept of conducting CFD analysis was to study the effect of air flow distribution inside the reactor, turbulence eddy dissipation, eddy viscosity and its effect on heat transfer. The CFD analysis was divided into 2 cases to get detailed understanding of flow pattern and heat transfer characteristics.

**Case 1: Simulation without Biomass**
In 1° case (length 1200 mm and dia. 275 mm) eddy viscosity (Y axis) was plotted against turbulence kinetic energy (X axis). The vector plot shows that maximum velocity could be found at air inlet 3.5 m/sec which is supplied at bottom right portion of reactor (Fig.5). The velocity vector also shows air flow direction which is perpendicular to divergent nozzle (Fig.6).

The multiple region of swirls generated shows eddy formation processing turbulence kinetic energy in reactor (Fig. 7, 8 & 9). Due to turbulence retention time, generated Producer gas would be more. Apart from that it will also help to improve the quality of Producer gas in reactor. Effect of reactor dimensions on turbulence kinetic energy (TKE) and Turbulence Eddy dissipation (TED) is also calculated with the help of CFX tools of ANSYS (Table 2).

<table>
<thead>
<tr>
<th>Design</th>
<th>Reactor dimension in mm</th>
<th>T K E, m² s⁻¹</th>
<th>TED, m² s⁻³</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case 1</td>
<td>L=1000, D= 275</td>
<td>0.00880</td>
<td>0.53067</td>
</tr>
<tr>
<td>Case 2</td>
<td>L=1200, D= 275</td>
<td>0.00921</td>
<td>0.20317</td>
</tr>
</tbody>
</table>

Critical analysis of Table 2 clearly indicates that higher values of turbulence kinetic energy could be obtained for bigger length of reactor. Higher values of turbulence kinetic energy help to improve Producer gas retention time in a reactor and hence produce clean gas due to thermal cracking of hydrocarbon.

**Case 2: Simulation with Biomass**
The horizontal biomass gasifier reactor model was set to thermal energy and four chemical reactions (R₁, R₂, R₃, and
R4) at filled reactor conditions. Each reaction having specific reaction rates is shown in Fig. 11 & 12 and Table 3.

Table 3: The reaction rates considered for simulation

<table>
<thead>
<tr>
<th>REACTION</th>
<th>REACTION DETAILS</th>
<th>REACTION RATE (mol m(^{-3}) s(^{-1}))</th>
</tr>
</thead>
<tbody>
<tr>
<td>R(_1)</td>
<td>C + O(_2) = CO(_2)</td>
<td>10e³</td>
</tr>
<tr>
<td>R(_2)</td>
<td>C + CO(_2) = 2CO</td>
<td>100</td>
</tr>
<tr>
<td>R(_3)</td>
<td>C + H(_2)O = CO + H(_2)</td>
<td>10e³</td>
</tr>
<tr>
<td>R(_4)</td>
<td>C + H(_2) = CH(_4)</td>
<td>3e⁻³</td>
</tr>
</tbody>
</table>

Close look of figures indicate that the maximum temperature in both the designs (reactor dimensions 1200 mm length, 275 mm Dia. and 1000 mm length, 275 mm Dia.) can be obtained nearer to air inlet. As we move towards outlet of reactor the temperature starts decreasing. This is due to higher mass fraction of oxygen near the inlet. The mass fraction of oxygen started reducing once we move away from inlet, which results in decreasing temperature. The temperature profile along length of chamber moving towards exit is shown in Figure 13.

To estimate the energy content of generated fluid with per unit mass static enthalpy for both the designs was calculated using the equation

\[ dh = c_p dT + \frac{1}{\rho} \left[ 1 + \frac{T}{\rho} \left( \frac{\partial \rho}{\partial T} \right)_p \right] d\rho \]  

(6)

Where, \(c_p\) is specific heat and \(\partial \rho / \partial T\) is density of fluid (Fig. 14 & 15). As depicted from contour plots that enthalpy is more near biomass inlet (red line in contours) (Fig. 14).

Fig. 13: Temperature profile along with Z-axis for L=1200 mm & D=275mm (red line) and L=1000 mm, D=275mm (blue line)

Fig. 14: Temperature profile for L=1200mm, D=275mm

Fig. 15: Eddy viscosity for L=1200mm, D=275mm

Fig. 15 shows the eddy viscosity behavior with the constant inflow of biomass to the reactor as compared to without inflow of biomass. Close look of Fig. 15 clearly indicate that with the continuous reactions and inflow of biomass the eddy dissipation value increases along with zones. High eddy dissipation is formed near inlet and concentration eddy dissipation start diluting once we move towards outlet of reactor. Composition of generated gas was also studied form
the same software and its results are shown in Figures 16 to 19.

Critical analysis of Figures indicate that as we move towards exit of reactor the molar mass concentration of oxygen reduces (Fig.19). This contributes to decreasing of temperature as depicted from figure 13. The reduction in oxygen has direct affect on reduction of gasification reaction and hence composition of producer gas.

VI. CONCLUSION

From the above simulation study following conclusions can be drawn:
1. All the reactions take in zones and largely depend on static enthalpy.
2. The oxidation zone is predominant above air inlet.
3. Temperature distribution is non-uniform along the length of chamber moving towards gas outlet from reactor.
4. The reactor dimensions, length 1200 mm and diameter 275 mm was found better in terms of temperature profile and gasification reaction.
5. Turbulence kinetic energy (TKE) (0.00921 m² s⁻²) and Turbulence Eddy dissipation (TED) (0.20317 m² s⁻³) is found better for the higher length reactor (length 1200 mm x 275 mm φ) compared to smaller length reactor (length 1000 mm x 275 mm φ). For smaller reactor these figures are 0.00880 m² s⁻² and 0.53067 m² s⁻³ respectively.

REFERENCES


