International Journal of Current Engineering and Technology
ISSN 2277 - 4106
© 2013 INPRESSCO. All Rights Reserved.
Available at http://inpressco.com/category/ijcet

Research Article

CFD Simulation of 20 KW Down Draft Gasifier

C. Mohan Raj[a] and P.K. Srividhya[b*]

[a]Deptt. of Aeronatical,Periyar Maniammai University, Vallam, Thanjavur
[b]Deptt. of Mechanical Engg, University, Vallam, Thanjavur

Accepted 22 Feb. 2013, Available online 1March 2013, **Vol.3, No.1 (March 2013)**

Abstract

The gassifier unit at Periyar Maniammai University, India has been active since 2003 using the CFD Simulation the performance analysis were studied using the 20 Kw Down Draft Thermo gasifier. The complete model was created in GAMBIT and simulated in ANSYS FLUENT. The entire simulation work was done through species transport phenomena. The fluent coal model was modified based on proximate and ultimate analysis of wood. The process parameters were bulk density of wood chips (porosity) and velocity of oxidizer inlet. The porosity was varied at 0.5 25, 0.485 and 0.439.At porosity 0.525, efficient temperature distribution was obtained. Also the oxidizer velocity was varied between 3m/s and 13 m/s (3, 4, 5, 7,9,11 and 13m/s).The maximum temperature reached in gasifier was increased marginally with incr ease of oxidizer velocity. But the temperature distribution was not good enough with increase in oxidizer velocity. It was found that there is more amount of unburned fuel at velocities 11m/s and 13m/s.

Keywords: *Gasification; CFD simulation; Bulk density; Oxidizer velocity.*

1. Introduction

Due to increase in energy demand and fossil fuel depletion the alternate sources such as wind, solar, biomass, fuel cell were being increasingly utilized. For 20 years, the biomass was one of the most relied renewable energy sources in the world. Biomass is organic material from agricultural and animal wastes. Biomass research also indicated that direct combustion and gasification techniques were applicable to generate energy supplies for future. Periyar Maniammai University had already involved in this area. A 200 kW Biomass gasifier for power production and 20 kW thermal gasifier plant were available for carrying out research. The current work was focused on computational modeling of fixed bed downdraft 20 kW gasifier in Periyar Maniammai University.

Marta Muilerburg *et al*, investigated the CFD modeling of combustion zone. They have varied the bulk density, inlet velocity of oxidizer and fuel types. Changing porosity in combustion and gasification zones indicated that a fuel with a packing density that correlates to a porosity of 0.5 would be the best. Oxidizer velocity of 15m/s showed more complex combustion and 1 m/s provides larger combustion area with un burnt fuels. When compared the wood chips and corn, the corn had smaller area of unburnt fuel and wood had larger area of unburnt fuels. Fletcher *et al* , modeled biomass particulate via Lagrangian approach which provides detailed information on gas composition and temperature at the outlet. Different operating scenarios were efficiently examined. The model was a powerful tool and validation against detailed experimental data would aid with design process of gasifiers. Mermourd *et al*, proposed an analysis of thermo chemical situation and demonstrated that only the

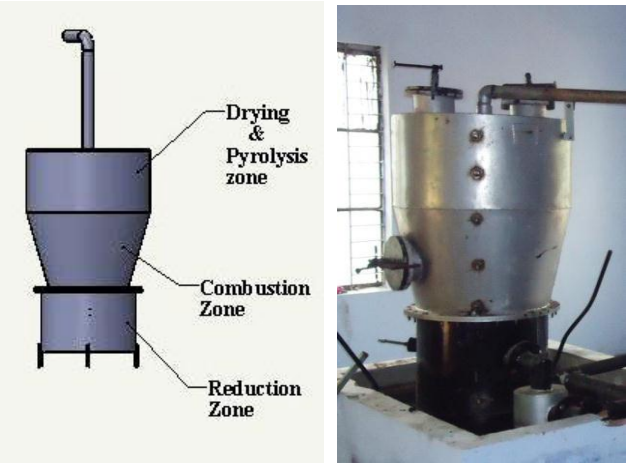


Figure. 1, 20 kW Gasifier model

assumption of thermal equilibrium between the particle and the surrounding gas was valid for a model at bed scale. A numerical model based on balance equations for gas species and enthalpy and including heterogeneous reaction kinetics, was developed where they were able to retrieve both qualitatively and quantitatively with an accuracy of 7% until 60% of the conversion. It was possible to characterize the thermo chemical situation.

*Corresponding author: P.K. Srividhya

Martin Miltner *et al*, aimed to maximize the thermal efficiency and reduce gaseous and particulate matter emissions. The presented model for the prediction of NOx-emissions showed very good results in a wide range of the overall equivalence ratio. Yigun Wang *et al* , studied the thermo chemical conversion of biomass using CFD. They concluded that CFD would be used as a powerful tool to predict biomass thermo chemical processes as well as to design thermo chemical reactors. Martin Miltner *et al* , investigated to maximize the thermal output of the combustion and to minimize the gaseous emissions. The flue gas recirculation rate and the overall equivalence ratio could be varied in order to reduce gaseous emissions of CO and NOx. Examination of the predicted biomass bale burnout could be a great help for maximization of the thermal output of the combustor. From the earlier investigations, it could be concluded that a very few researchers had done work on the DFD simulation.

2. Modeling

Modeling together with experiments enable a cost-effective approach for future biomass combustion application design. The most advanced modeling tools ware CFD tools and the researchers used fluent as the software for CFD.

The two dimensional 20kW downdraft gasifier was modeled in GAMBIT software package and exported to FLUENT solver.

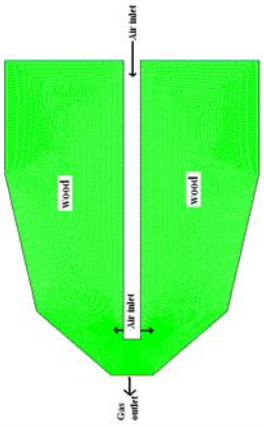


Figure. 2, 2-D Computational gasifier model

The following were the general assumptions made in the study:
i) The flow was steady.
ii) No-slip condition on wall surfaces.
iii) Chemical reaction was faster than the time scale of the turbulence eddies.

Governing Equations

The equations for conservation of mass, conservation of momentum, and energy equation were given as:

$$\nabla.(\rho\, v) = S_m \quad (1)$$

$$\nabla.(\rho vv) = \rho g - \nabla p + \nabla.(\tau) + S_f \quad (2)$$

$$\nabla.\big(v(\rho E + p)\big) = \nabla.K\nabla T - \nabla.\sum hi\, Ji + \nabla.(\tau.v) + S_h \quad (3)$$

Where K is the effective conductivity and *Ji* is the diffusion of species i.

The first three terms on the right hand side of equation **(3)** represent heat transfer due to conduction, species diffusion, and viscous dissipation. S_h is a source term including the enthalpy formation from the chemical reaction of the species. The energy E defined as

$$E = i + \frac{1}{2}(u^2 + v^2 + w^2)$$

Where *i* is internal energy

$$E = h - \frac{P}{\rho} + \frac{1}{2}(u^2 + v^2 + w^2) \quad (4)$$

Where *h is* the sensible enthalpy and it is given as

$$h = i + \frac{p}{\rho}$$

$$i = c_p T$$

$$h = \int_{T_{ref}}^{T} C_{p\,i}\, dT$$

T_{ref} = 298.15 K.

Turbulence model

The standard K-ε model was considered for the simulation. It was simplest 2-equation model and suited for the well bounded cases. This turbulence model was based on turbulent kinetic energy k and dissipation rate ε. The turbulent kinetic energy and dissipation rate had been obtained from following equations,

$$\frac{\partial(\rho k)}{\partial t} + div(\rho kU) = div\left[\frac{\mu_t}{\sigma_k} grad\ k\right] + 2\mu_t E_{ij} - \rho\varepsilon \quad (5)$$

and

$$\frac{\partial(\rho \varepsilon)}{\partial t} + div(\rho \varepsilon U) = div\left[\frac{\mu_t}{\sigma_\varepsilon} grad\ \varepsilon\right] + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij}.E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad (6)$$

Where the eddy viscosity $\mu_t = \rho\ C_\mu \frac{k^2}{\varepsilon}$; C_μ was dimensionless constant

The values for the adjustable constants are

C_μ = 0.09; σ_k = 1.00; σ_ε = 1.30; $C_{1\varepsilon}$ = 1.44; $C_{2\varepsilon}$ = 1.92

The turbulence models were valid for the core flow i.e. the flow in regions far from walls. The flow near to the walls was affected by the presence of walls. Here viscous damping and kinematic blocking which reduce the tangential velocity fluctuations and normal fluctuations respectively were considered. The solution in near to the wall region is very important because the variables had large gradients. So wall function which was collection of semi empirical formulas and functions instead of considering boundary layer study was considered.

Radiation model

P1 radiation model for calculating radiation flux inside the gasifier was considered here.

$$-\nabla q_r = aG - 4aG\sigma T^4$$
$$q_r = -\frac{1}{3(1+\sigma_s)-C\sigma_s}\nabla G \quad (7)$$

and q_r the radiation heat flux, a the absorption coefficient, σ_s the scattering coefficient, G is the incident radiation, C the linear-anisotropic phase function coefficient, and s is the Stefan-Boltzmann constant.
The radiation flux at the walls

$$q_{r(w)} = \frac{4\pi\varepsilon_W\frac{\sigma T_W^4}{\pi}-(1-\rho_w)G_w}{2(1+\rho_W)} \quad (8)$$
$$\varepsilon_W = 1-\rho_w$$

Where ρ_w the wall reflectivity

Combustion model

The global reaction mechanism was modeled to involve the following chemical species: C, O_2, N_2, CO, CO_2, H_2O and H_2. All of these species were assumed to mix in the molecular level. The chemical reactions inside the gasifier were modeled by calculating the transport and mixing of the chemical species by solving the conservation equations describing convection, diffusion, and reaction of each component species. While choosing the equation to solve conservation equations for chemical species, FLUENT selected the local mass fraction of each species, Yi, through the solution of a convection-diffusion equation for the i^{th} species. This conservation equation took the following general form:

$$\frac{\partial}{\partial t}(\rho Y_i) + \nabla.(\rho U Y_i) = -\nabla . J_i + R_i + S_i \quad (9)$$

where R_i the net rate of production of species, i by chemical reaction and S_i is the rate of creation by addition from the dispersed phase plus any user-defined sources. An equation of this form would be solved for species N-1 species where N is the total number of fluid phase chemical species present in the system. Since the mass fraction of the species must sum to unity, the N^{th} mass fraction is determined as one minus the sum of the N - 1 solved mass fractions. To minimize numerical error, the N^{th} species should be selected as that species with the overall largest mass fraction, such as N_2 when the oxidizer was air.
The reactions needed to be solved

$C + O_2 = CO_2$ (1)
$2H_2 + O_2 = 2H_2O$ (2)
$C + CO_2 = 2CO$ (3)
$C + H_2O = CO + H_2$ (4)
$CO + H_2O = CO_2 + H_2$ (5)
$C + 2H_2 = CH_4$ (6)
$CO_2 + H_2 = CO + H_2O$ (7)

The assumption in this model was that the chemical reaction was faster than the time scale of the turbulence eddies (Eddy-dissipation model). Thus, the reaction rate was determined by the turbulence mixing of the species. The reaction was assumed to occur instantaneously when the reactants meet.

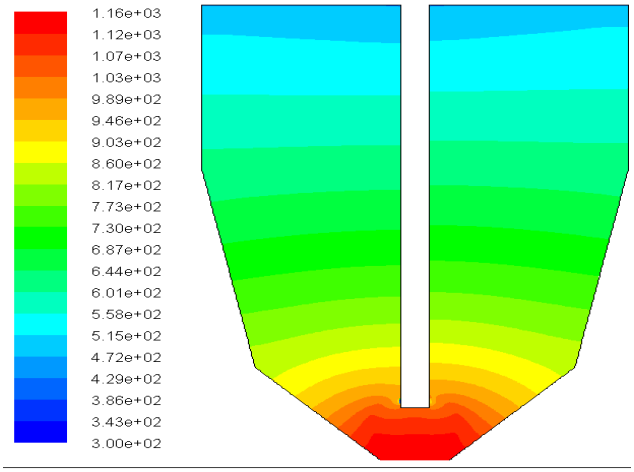
Boundary types

Boundary types were very important for solving the CFD problems. All the air inlet surfaces were defined as mass flow rate inlets. The mass flow rate, temperature of the mixture, mass fractions of all species in wood volatiles, turbulent intensity and hydraulic diameter are specified. Mass flow inlets were indented for compressible flows than incompressible flows. It was more difficult to converge the pressure inlets. The outlet surface was assigned as a pressure outlet boundary. The pressure, temperature, turbulent intensity and hydraulic diameter were specified. This did not affect the calculations inside the computational domain but would be used if backflow occurs at the outlet. It was suitable for both compressible and incompressible flows. All back flow quantities could be specified but in this case back flow effects were neglected. For ideal gas flow, non-reflecting outlet boundary conditions were available. The outside surfaces were defined as wall boundary. The walls were stationary with no-slip condition imposed (zero velocity) on the surface. For adiabatic case, the heat flux on the wall was set to 0 (zero). For constant wall temperature, the wall temperature is set to a certain constant value.

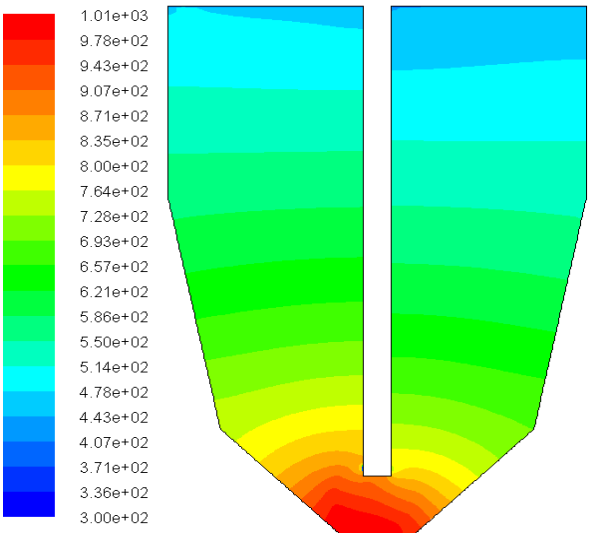
Before FLUENT can begin solving governing equations, flow field guessed initial values, used as the initial values of the solution, had to be provided. Once the initial values had been provided, the iteration was performed until a converged result is obtained.

3. Results and discussions

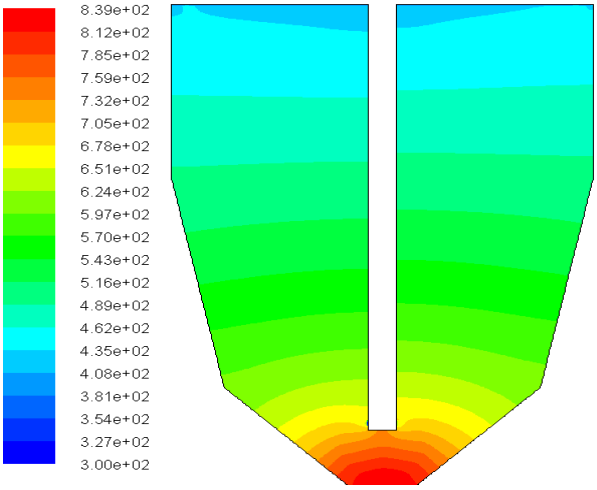
Temperature distribution of gasifier varies with porosity



Contours of Static Temperature (k)
Porosity = 0.525



Porosity = 0.483



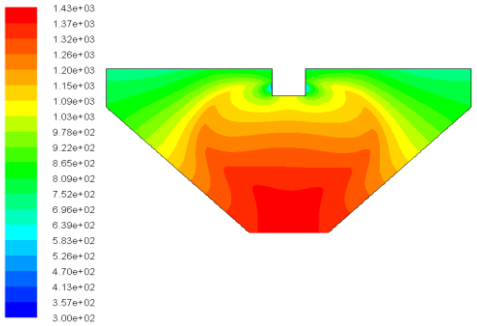
Porosity = 0.439

Figure 3, Temperature contours with varies porosities

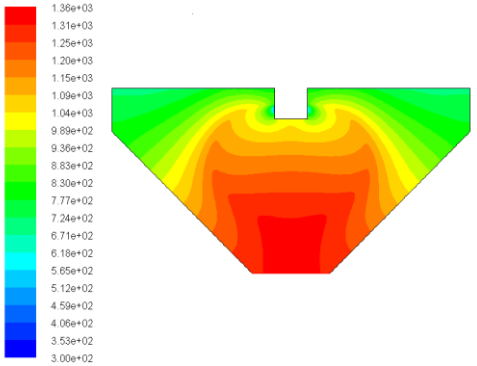
The particle size was an important parameter in biomass gasification because it determined the bed porosity and thus the fluid-dynamic characteristics of the bed. On the other hand, fine grained fuels led to substantial pressure drops in fixed bed reactors. Temperature variations due to porosity had been shown in above figures. All these cases show the highest temperature at the combustion and reduction zones. At porosity 0.525, the maximum temperature reached was 1116K. At the air inlet, the temperature reached ranges between (1112-1116) K. At the combustion zone, the temperature ranges between (1003-1007) K. At porosity 0.4831, it was seen that there was instable temperature distribution. The maximum temperature at the reduction zone ranged between (978-1001) K and at combustion zone, it ranged between (943-978) K. At porosity 0.439, the maximum temperature distribution is concentrated more on the reduction zone that ranged between (812-839) K. The temperature ranges between (651-732) K at combustion zone.

The porosity 0.525 had the event and better temperature distribution throughout the porous zone as shown in figure 3. And also temperature reached higher when compared with other conditions.

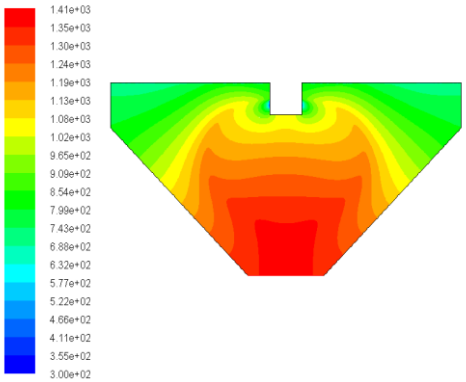
Temperature distribution of gasifier varies with inlet oxidizer velocity



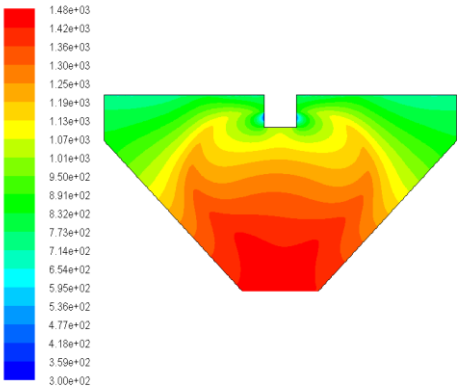
3m/s



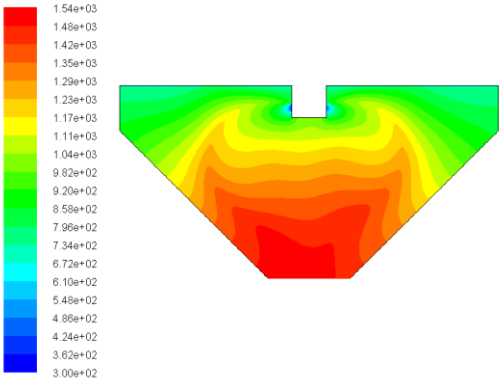
4m/s



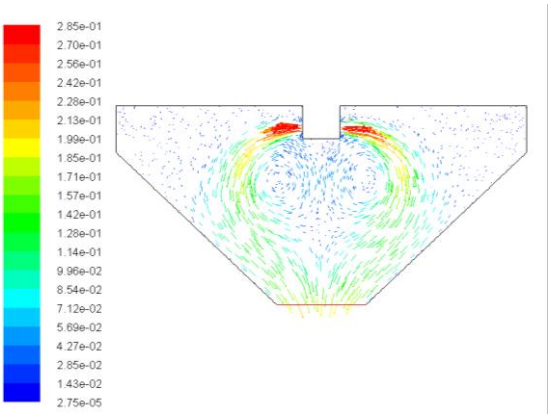
5m/s



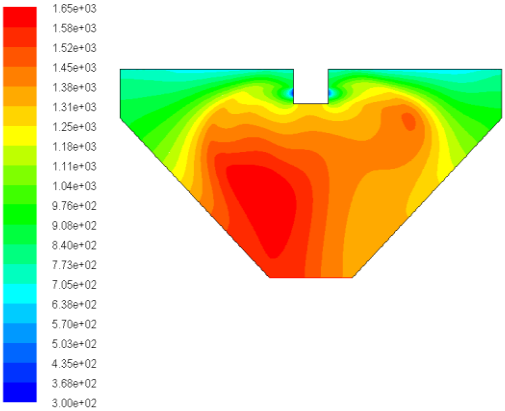
7m/s



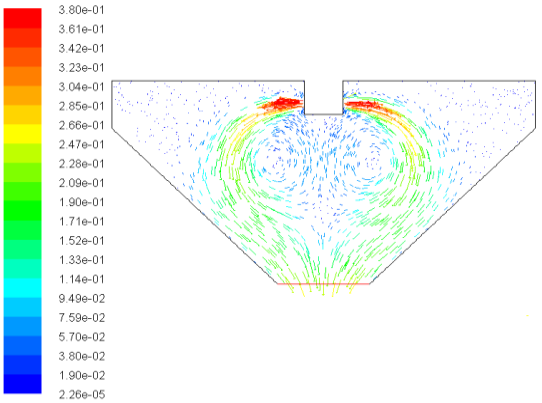
9m/s



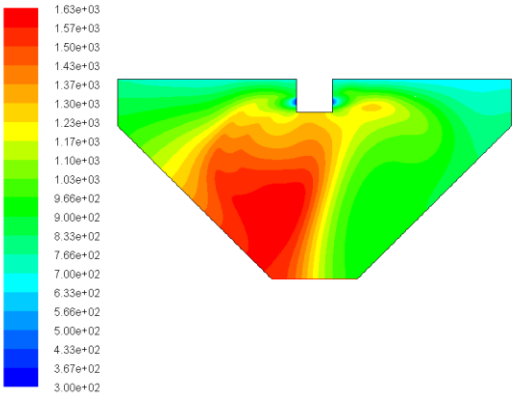
3m/s



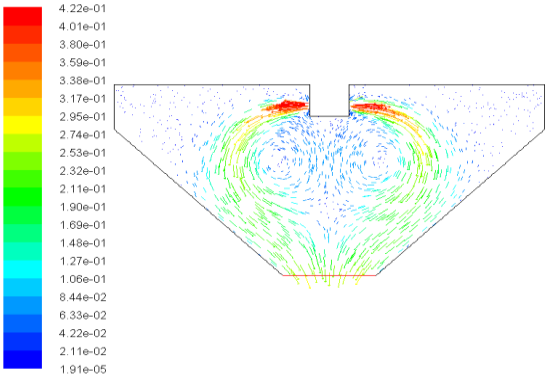
11m/s



4m/s



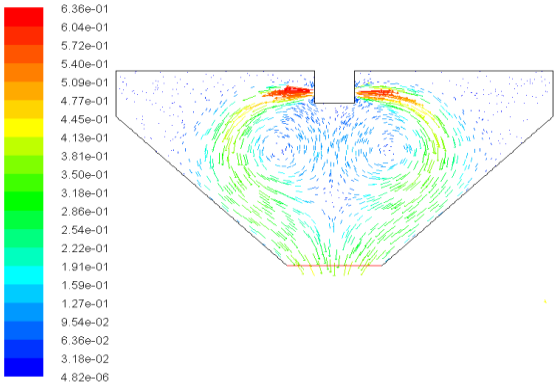
13m/s



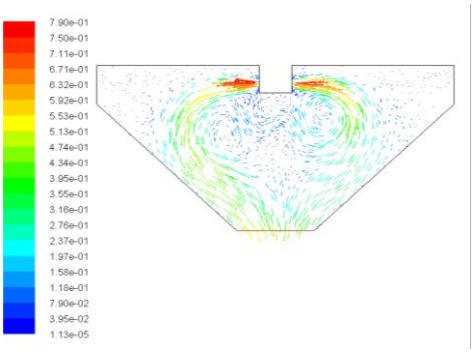
5m/s

Figure 4, Temperature contours with varies oxidizer velocities

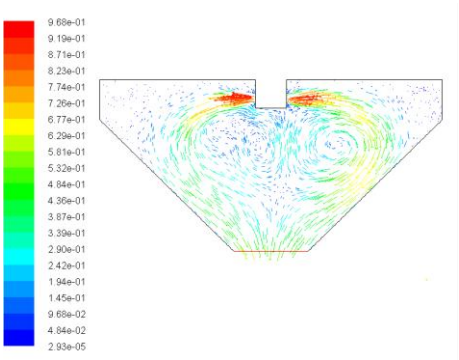
The reduction zone of gasifier was simulated with various oxidizer velocities. The results of temperature contours had been shown in figure 4. From the simulation, it was found that temperature fluctuated with increased velocities. Another important parameter was the amount of unburnt fuel presence inside the gasifier was increased with increasing oxidizer velocities. At oxidizer velocity 9 m/s,11 m/s and 13 m/s, it was found that there was more unburnt fuel in reduction zone. Also from velocity 11 m/s, the combustion instability was started for this model.



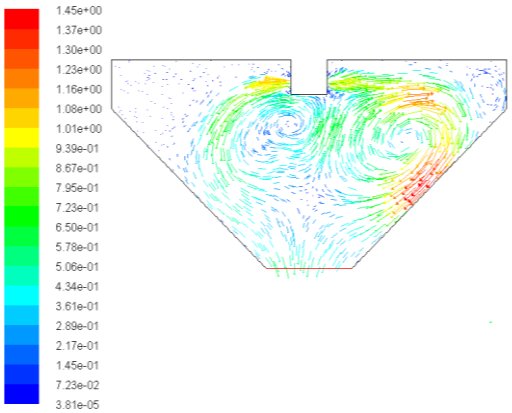
7m/s



9m/s



11m/s



13m/s

Figure 5, Velocity vectors for varies oxidizer velocities

The oxidizer velocity was a main parameter for the biomass combustion. The oxidizer velocity vector contours were shown in figure 5. The shape and position of vortex in inside gasifier was maintained upto 7 m/s. But from velocity 9 m/s, the shape and position of vortex had varied. This might lead to combustion instability.

4. Conclusion

The main objective of this study is to find the optimum value of porosity and oxidizer velocity through CFD simulation for 20kW gasifier model. For this study porosity of 0.525, 0.483 and 0.439 were chosen. It was found at 0.525, the temperature reached its highest value and also there was more even temperature distribution. The oxidizer velocity was varied between 3 m/s and 13 m/s for simulation. It was found that at velocities 4 m/s and 5 m/s, the combustion reached higher temperature value resulting in very less amount of unburnt fuel presence inside gasifier and more combustion stability. But higher than 9 m/s, the amount of unburnt fuel inside gasifier was increased and combustion instability was introduced.

References

Marta Muilenburg, Yunye Shi, and Albert Ratner. November 11-17(2011) ,Computational modeling of the combustion and gasification zones in a downdraft gasifier,. *Proceedings of the ASME 2011 International Mechanical Engineering Congress & Exposition IMECE2011*., Denver, Colorado, Colorado, USA.

D.F. Fletcher, B.S. Haynes, F.C. Christo, S.D. Joseph (2000) ,A CFD based combustion model of an entrained flow biomass gasifier, *Journal of Applied Mathematical Modelling*, 24 , 165-182

F. Mermoud, F.Golfier, S. Salvador, L. Van de Steene, J.L. Dirion (2006) ,Experimental and numerical study of steam gasification of a single charcoal particle, *Journal of Combustion and Flame* 145 ,59–79

Martin Miltner, Angela Miltner, Michael Harasek, Anton Friedl (2007) ,Process simulation and CFD calculations for the development of an innovative baled biomass-fired combustion chamber, Journal of Applied Thermal Engineering 27 ,1138–1143

Yiqun Wang and Lifeng Yan (2008) ,CFD Studies on Biomass Thermochemical Conversion, *International Journal of Molecular Sciences, 9*, 1108-1130

Anil K.Rajvanshi. Published as a Chapter (No. 4) in Book '' Alternative Energy in Agriculture'' Vol. II, Ed. D. Yogi Goswami, CRC Press, 1986

Anjieddy Bhavanam And R. C. Sastry (December 2011) ,Biomass Gasification Processes in Downdraft Fixed Bed Reactors,: A Review. *International Journal of Chemical Engineering and Applications*, Vol. 2, No. 6,

B.V. Babu* and Pratik N. Sheth ,Modeling and Simulation of Reduction Zone of Downdraft Biomass Gasifier: Effect of Air to Fuel Ration,, B.I.T.S., PILANI-333031 (Rajasthan) INDIA.

S.Sivakumar*, K.Pitchandi, and E.Natarajan. ,Design and of Down Draft Biomass Gasifier using computational Fluid Dynamics., Department of Mechanical Engineering, College of Engineering, Guindy, Anna University, Chennai-25. INDIA

D.S.Mandwe, S.R.Gadge, A.K.Dubey, and V.P.Khambaikar.(November 2006) ,Design and development of a 20 kW cleaning and cooling system a wood-chip gasifier,. Journal of Energy in Southern Africa Vol 17 No 4.

Rajiv Varshney, Bhagoria.J.L, Mehta.C.R.(2011) ,Experimental investigation on a biomass briquette based throat less downdraft gasifiers,. *International journal of applied engineering research, dindigul* Volume 2,

Pratik N Sheth and B V Babu. ,Modeling and Simulation of Downdraft Biomass Gasifiers,. Birla Institute of Technology and Science (BITS), PILANI-333031 (Rajasthan) INDIA.

Michacl J. Bockelie Martin k. Denison, Zumao chen, Temi Linjewile, Constance L. Senior, Adel F. Sarofim. ,CFD Modeling For Entrained Flow Gasifiers,. Reaction Engineering International, 77 West 200 South, Suite 210, Salt Lake City, UT 84101.

Rathod V.P, RatnadhariyajJ.K. and Channiwala S.A. ,Design and development of downdraft venturi type gasifier,, surat-395007 Gujarat state, India.

K.Sivakumar and N. Krishna Mohan (Jan 2010) ,Performance analysis of downdraft gasifiers for agriwaste biomass material,. *Indian Journal Science and Technology*. Vol.3 No 1

B. V. Babu, Pratik N. Sheth. ,Modeling and Simulation of reduction Zone of downdraft biomass gasifier; Effect of Char reactivity factor,. Department of Chemical Engineering, Birla Institute of Technology and science (BITS), Pilani Vidya Vihar Campus, Pilani 333031, Rajasthan, India.

S. Sivakumar, K. Pitchandi, and E.Natarajan.(2008) ,Modeling and Simulation of Down Draft Wood gasifier,. *Journal of Applied Sciences* 8 (2): 271-279.