# Numerical Study of Downdraft Gasifier

Mohammad Husain Shaikh[1] , Samirkumar Narendrabhai Sinhal[2]

[1] *Assistant Professor, Department of Mechanical Engineering, Chhotubhai Gopalbhai Patel Institute of Technology, Bardoli, Surat*

[2] *PG Student, Department of Mechanical Engineering, Chhotubhai Gopalbhai Patel Institute of Technology, Bardoli, Surat*

*Abstract-* **This paper contains detailed CFD study of downdraft gasifier via simulating flow and reaction. The model is based upon the ANSYS Fluent package which represent powerful tool and can be used in gasifier design and analysis. 2-D geometry is created in ANSYS design modeller. Meshing is done with minimum mesh quality 80%. Using a k-epsilon turbulence model with integrated energy and continuity equation whole analysis is done will results into at 0.4 equivalence ratio and mole fraction of different species %CO, %CO2, H2 and N2 is calculated and validated present design with experimentation. Model provides detailed information of gas composition and temperature profile throughout gasifier at different operating condition of air flow rates to be examined in efficient manner.**

*Index Terms-* **Biomass, Biomass pallets, CFD Modelling, Gasification, Renewable Energy.**

## 1. INTRODUCTION

The drive to reduce net greenhouse emissions has produced considerable interest in the combustion of biomass. Many processes produce biomass waste that can be used for energy production.[1] This paper is concerned with the design and optimization of Downdraft Gasifier System. Our aim is to use Computational Fluid Dynamics (CFD) to prove the design of a pilot plant which is being developed, and then to use the validated model to produce higher capacity gasifier system.

In downdraft gasifier the fuel (Biomass) stored into gasifier and limited quantity of air is supplied centrally into gasifier via air inlet pipe. The quantity of air is limited, so that pyrolysis process takes place inside gasifier and leaves behind a mixture of CO, $H_2$, $CO_2$, $H_2O$, $CH_4$ and $N_2$. Once the volatiles released char and ash remains. Then char undergoes reaction with $CO_2$ and $H_2O$ to produce CO and $H_2$.

CFD modelling techniques are becoming widespread in the biomass thermo chemical conversion area. Researchers have been using CFD to simulate and analyze the performance of thermo chemical conversion equipment such as fluidized beds, fixed beds, combustion furnaces, firing boilers, rotating cones and rotary kilns. CFD programs predict not only fluid flow behaviour, but also heat and mass transfer, chemical reactions (e.g. devolatilization, combustion), phase changes (e.g. vapour in drying, melting in slagging), and mechanical movement (e.g. rotating cone reactor). Compared to the experimental data, CFD model results are capable of predicting qualitative information and in many cases accurate quantitative information. CFD modelling has established itself as a powerful tool for the development of new ideas and technologies. However, CFD modelling for biomass thermo chemical conversion still face significant challenges due to the complexity of the biomass feedstock and the thermo chemical process. Biomass is a mixture of hemicelluloses, cellulose, lignin and minor amounts of other organics with proportion and chemical structure affected by variety. Inorganic ash is also part of the biomass composition. The complex structure makes biomass compositions pyrolyzed or degrade at different rates by different mechanisms and affect each other during thermo chemical process, and it makes the biomass particle feedstock has anisotropic properties in physical characterization.[2]

A CFD model for the combustion zone was previously developed and validated with the experimental data got from the DTU 100 $kW^{th}$ two-stage gasifier. It includes detailed chemical mechanism with tar cracking, heat transfer with radiation and turbulent fluid flow. It was used for a brief study of sensitivity for different parameters that enlightened the influence of the air injection on tar cracking.[3]

Nevertheless, more research is needed because gasification depends always on the raw material characteristics and the reactor design. This work aims to simulate the thermal performance of a lab-scale downdraft fixed bed gasifier by means of an Euler-Euler multiphase CFD model, and validate it in accordance with experimental data of wood (biomass sawdust pallets) gasification.

## II. MODELLING OF DOWNDRAFT GASIFIER

The model chosen for this research focused on the combustion zone of biomass in a down-draft gasifier. The combustion zone would determine the temperatures in the gasifier and the reactions in the other zones and is therefore the pivotal zone in the gasification process. The geometry of the Downdraft Gasifier used in the simulations was generated using ANSYS Design Modeler. All the dimensions of the gasifier are according to final design. The schematic diagram of the gasifier considered in this study is shown in Fig 1. The entire height of the gasifier is 1642 mm. The biomass inlet and gas outlet diameters are 230mm and 30mm respectively. The combustion, pyrolysis and drying, and reduction zones have heights of 200 mm, 550mm and 230mm respectively. The long air inlet is having diameter 30 mm. ANSYS 18.0 was used to create a two-dimensional model of gasifier to be used in large scale testing. ANSYS FLUENT was then used to set the parameters of the model. The design has been taken from [4] and modified from the actual design of the gasifier for simplification in processing and to correct for combustion at the oxidizer input. Final design with the entire dimension is shown in figure 1. Whole study is 2D analysis study. The air nozzle is used to pass the air and produced gas will leaves through the gas outlet and the feedstock is charged through the biomass inlet. The downdraft biomass gasifier is modeled in Design Modeler and analysis is done by FLUENT. To avoid complexity solution the following assumptions such as steady flow, adiabatic wall and turbulent eddies is considered.

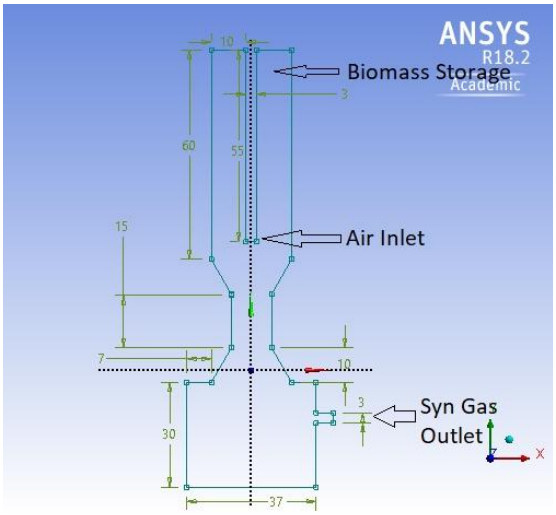

Figure 1 Designed of Downdraft Gasifier

Meshing and Boundary Conditions

The simulation grid was established on a 2D domain. The mesh was built consisting of triangle elements throughout the gasifier as exhibited in Fig. 2. Mesh quality was evaluated in accordance with squish, skewness, and aspect ratio criteria. Results of this evaluation indicated a maximum skewness 0.53, minimum orthogonal quality 0.78, maximum aspect ratio of 1.50. Statistics of meshing is No. of nodes 6993, No. of Elements 13150 with triangular shape and maximum face size is 0.007 m.

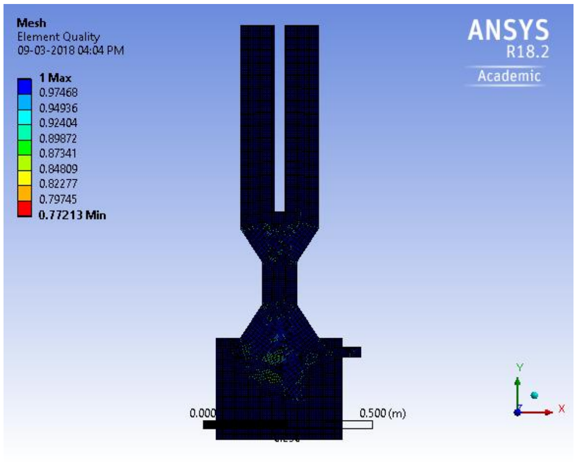

Figure 2 Meshing in Downdraft Gasifier

The boundary conditions established at the top of the domain, corresponding to the biomass entrance, were the raw material inlet velocity, wood composition, particle diameter and the initial temperature of wood. Thus, the raw material velocity was assumed equal to biomass consumption calculated in design section[4] temperature was similar to the ambient temperature, 300K. There was a 3cm diameter pipe located in the middle of the reactor where the air was introduced into the system. Here, some boundary conditions should be defined; the air mass flow rate corresponded to that one calculated at designed stage 0.001944 kg/sec.

Table 1 Lists of Boundary Condition

| | Name | Type | Information |
|---|---|---|---|
| Boundary condition | Fluid | Fluid | Air(21%$O_2$) |
| | Outlet-syngas | Exhaust Fan | - |
| | Inlet Air | Mass flow inlet | 0.00477 kg/s |
| | Wood inlet | Mass flow inlet | 0.001944 kg/s |
| | Wall | Insulated | No-slip |

At the bottom of the reactor, the mean temperature of the outlet gas was defined as the boundary conditions. Exhaust fan is considered as a boundary condition at outlet. The external vertical surface of the reactor was defined as a boundary of this process, and the conditions were established assuming this part as the wall. Hence, the boundary conditions on this limit included ambient temperature, 288 K, wall thickness, 0.15 m, the thermal conductivity of the wall and the insulator, 8 and 0.53 W/m$^2$.K and the external emissivity 0.1.

Model Formulation

The computational fluid dynamics CFD model was based on the Navier-Stokes equations of mass, momentum, and energy conservation, and coupled on a pressure-based transitory state. The ANSYS Fluent software solves these equations by default, but some issues might be defined; the model was established on a 2D axis-symmetric domain because the reactor had a cylindrical geometry. Mass, momentum and energy equation were analyzed in cylindrical coordinates taking into account the radial and axial directions. [5] The following were the general assumptions made in the study:[6]

a) The flow is steady and two-dimensional.

b) Wall surfaces at the no-slip condition.

c) The chemical reactions were faster than the time scale of the turbulence eddies.

d) Discrete phase model was used, given the small particle size and compared to the reactor volume.

e) The particles are spherical in shape and size 25mm has a uniform distribution.

f) The inner shell of gasifier considers for all chemical reaction.

Table 2 Lists of model settings

| | Model | Settings | Information |
|---|---|---|---|
| Models Settings | Space | 2D | - |
| | Time | Steady | - |
| | Viscous | Standard k-epsilon Turbulence model | Turbulence intensity = 10% |
| | Wall Treatment | Standard Wall Functions | - |
| | Specie Transport | Enabled | - |
| | Discrete Phase | Surface Injection | - |

*Turbulence model*

Most common RANS turbulence model applied to combustion is k-epsilon and its variants (standard and Realizable). The k-epsilon models provide a good solution without excessive computation time. So, here standard k-epsilon model with standard wall function is provided.

*Radiation model*

There are four common radiation models in Fluent; P-1 Radiation Model, Roseland Model, Discrete Ordinates, Model (DOM), Discrete Transfer Radiation Model (DTRM). P-1 Radiation Model is considered for this simulation process work. P-1 model is the simplest modified derivation of the P-N radiation model which is based on the expansion of the radiation intensity I into an orthogonal series of spherical harmonics and high order accuracy. [6]

*Species transport model*

The specie model is the best way for modeling of biomass/coal gasification. The species transport model has been chosen to model the chemical reactions inside the gasifier and to find out the composition of various species like CO, $CO_2$, $N_2$, $H_2$, and $CH_4$.

In species model non-premixed combustion is used for gasification with non-adiabatic energy treatment and empirical fuel stream. In model setting, fuel lower calorific value 20 MJ/kg, fuel specific heat 1760 J/kg.K, and fuel molecular weight 30 kg/kmol data is provided.

In boundary tab all species concentration in mass fraction is provided from the ultimate analysis data of wood. PDF table is generated in Table tab and then exported it into file format for future calculation.

*Discrete Phase*

Injection of wood particle in gasifier is created by this model. In this model Create injection with surface injection or single injection type with particle of combustion and material as a wood. Custom laws are marked in which inert heating, surface combustion, Devolatization laws are activated. In the point properties diameter, temperature, flow rate are provided with values 0.025m, 300K, 0.001944 kg/sec respectively. Thus injection of the wood particle is modeled.

Solution Methods

For solution of above problem methods used is tabulated in the table 3.

Table 3 Solution Methods and Discretization Scheme

| Solvers | Variable | Discretization scheme |
|---|---|---|
| | Pressure | PRESTO |
| | Momentum | Second Order Upwind |
| | Turbulent kinetic energy | Second Order Upwind |
| | Turbulent Dissipation Rate | Second Order Upwind |
| | Energy | Second Order Upwind |

## III. RESULTS AND DISCUSSION

Different parameter need to be calculated, which affects the composition of Syn-Gas like mass fraction of CO, $H_2$, $CO_2$, $H_2O$, $N_2$, $CH_4$. For equivalence ratio ranging from 0.3 to 0.7 these entire gases mass fraction contour is generated, also temperature contour is included.

Contour of Static Temperature

Temperature distribution within the downdraft gasifier inner shell at desired operating condition is given in temperature profile contour. Temperature profile for all the equivalence ratio from 0.3 to 0.7 is shown below from fig 3 to fig 7.

It has been seen from the contour that temperature inside gasifier is increased as equivalence ratio increases. That is because of more amount of oxygen available for combustion increase temperature of gasifier.

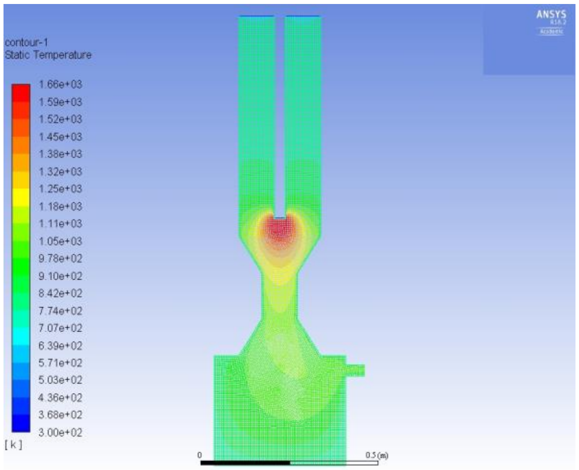

Figure 3 Temperature Contour for 0.3 ER

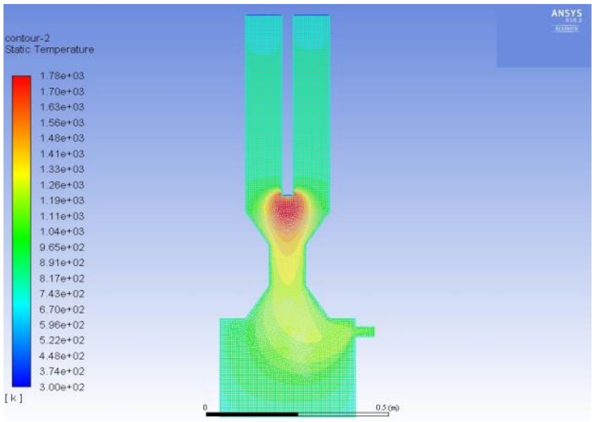

Figure 4 Temperature Contour for 0.4 ER

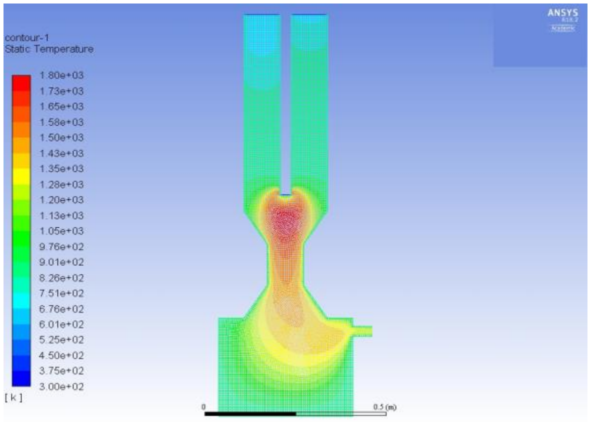

Figure 5 Temperature Contour for 0.5 ER

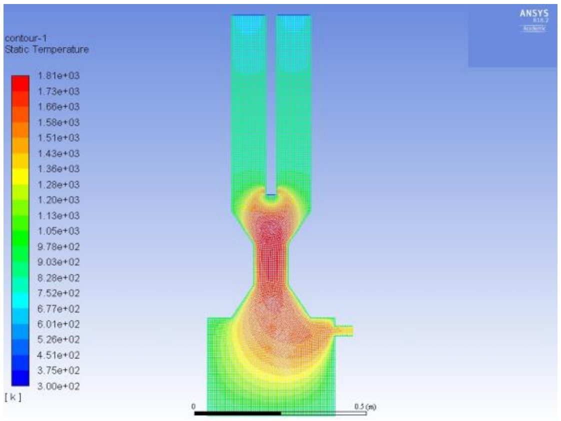

Figure 6 Temperature Contour for 0.6 ER

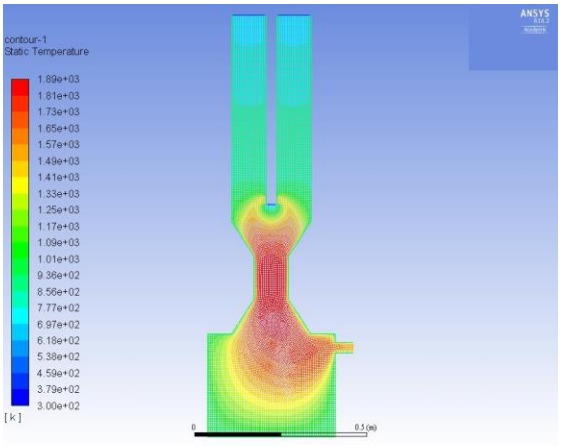

Figure 7 Temperature Contour for 0.7 ER

### Contour of mass fraction of CO

A fig. 8 and fig. 9 shows the mass fraction of Carbon Monoxide for range of equivalence ratio 0.4 & 0.5.

It is observed that carbon monoxide mass fraction decreases as ER increases and increase with decrease in ER. That is because of increases in ER can increases probability of combustion inside gasifier than gasification.

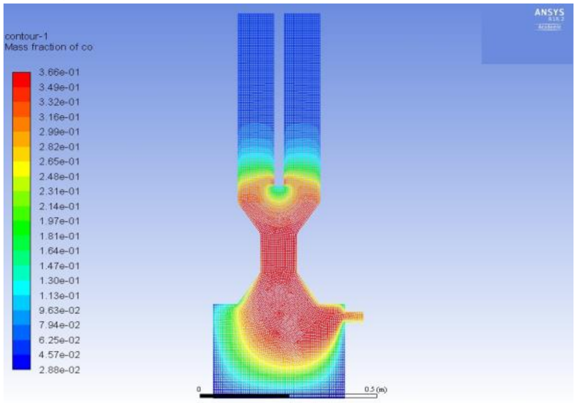

Figure 8 Mass Fraction of CO for 0.4 ER

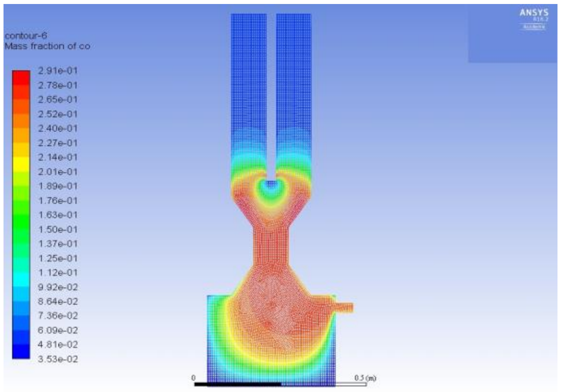

Figure 9 Mass Fraction of CO for 0.5 ER

### Contour of mass fraction of $H_2$

A figure 10 and figure 11 shows the mass fraction of Hydrogen for range of equivalence ratio 0.4 & 0.5.

It is observed that Hydrogen mass fraction decreases as ER increases and increase with decrease in ER.

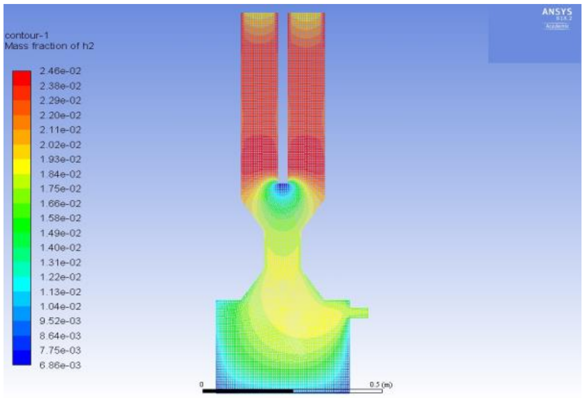

Figure 10 Mass Fraction of $H_2$ for 0.4 ER

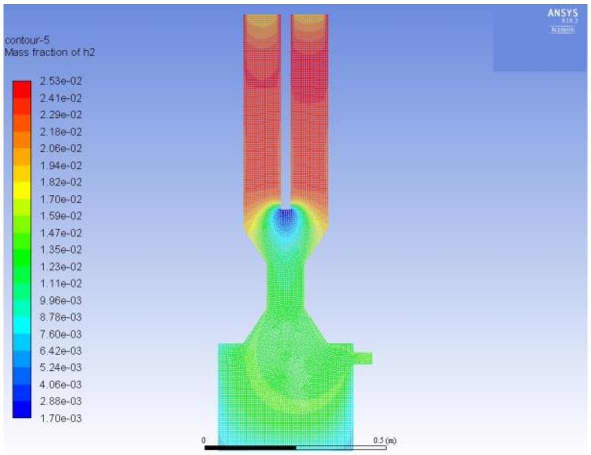

Figure 11 Mass Fraction of $H_2$ for 0.5 ER

### Comparison of ER and % Mass Fraction of Different Species

Figure 12 graph shows comparison between Elemental compositions of gases for different equivalence ratios. Best quality of gas we can get between 0.4 & 0.5 with maximum amount of CO & $H_2$.

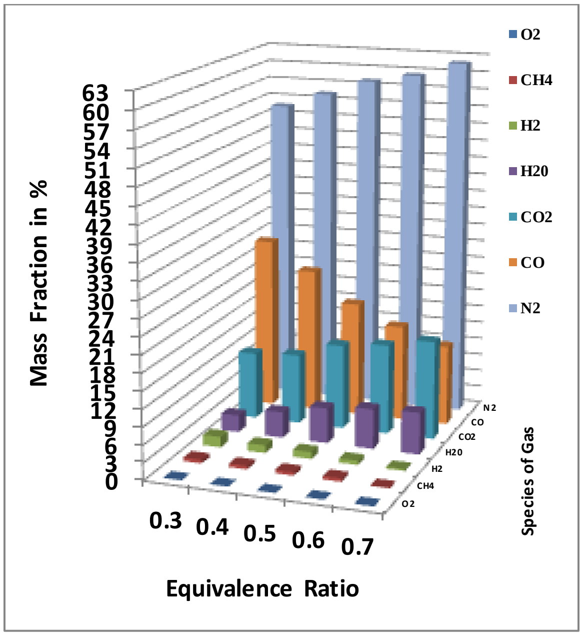

Figure 12 Comparison of ER and % Mass Fraction of Different Species

Gas Composition

Experiments were conducted at designed condition of air flow rate. For this condition gasifier is operated at equivalence ratio at designed value of 0.4. For this experimental condition equivalence ratio varies in between 0.4 to 0.5 Predicted gas quality for equivalence ratio 0.4 and 0.5 are given below in table 4.

Table 4 Predicted Gas Quality

| Equivalence ratio (Ø) | 0.4 | 0.5 |
|---|---|---|
| | Mass Fraction in % | |
| Hydrogen (H2) | 1.35 | 1.2 |
| Carbon Monoxide (CO) | 25.2 | 20 |
| Carbon Dioxide (CO2) | 12.2 | 14.8 |
| Water Vapor ($H_2O$) | 4.52 | 6.16 |
| Nitrogen ($N_2$) | 55.3 | 58 |

## IV. CONCLUSION

In this study, CFD modeling of fixed bed downdraft biomass gasifier has been conducted to get an innovative clean biomass gasification technology by using the commercial CFD solver ANSYS/FLUENT. A 2-D steady-state model was developed to simulate biomass gasification in a downdraft reactor. The standard k–ε turbulence model was used and equations of continuity, motion, and energy were integrated with the kinetics of homogeneous and heterogeneous reactions to calculate mass and energy transfer in the gasifier. The results yield comprehensive information concerning the thermal-flow behaviour and gasification process existing inside the specially designed fixed-bed downdraft gasifier. Based on the results obtained in the simulation from this study, the following conclusions are drawn.

For Various equivalence ratios the model has been tested and it was found that the mole fractions of combustible species decreased and the overall temperature in the gasifier increased as the values of ER rose in the selected range. Best quality of gas can be produce at equivalence ratio between 0.4 and 0.5.

## REFERENCES

[1] D. F. Fletcher and S. D. Joseph, “A CFD based combustion model of an entrained flow biomass gasifier,” vol. 24, no. 2000, pp. 165–182, 2006.

[2] C. Di Blasi, “Modeling chemical and physical processes of wood and biomass pyrolysis,” *Prog. Energy Combust. Sci.*, vol. 34, no. 1, pp. 47–90, 2008.

[3] L. Gerun and M. Tazerout, “Numerical Investigation of Air Injection During Biomass Gasification in a Two-Stage Downdraft Biomass Gasification in a Two-Stage Downdraft,” 2005.

[4] S. S. Narendrabhai, “Design , development and experimental studies of downdraft gasifier,” vol. 4, no. 3, pp. 1279–1285.

[5] C. A. Forero-Núñez, S. Ramirez-Rubio, and F. E. Sierra-Vargas, “Analysis of charcoal gasification on a downdraft fixed bed gasifier by CFD modeling,” *Int. Rev. Mech. Eng.*, vol. 9, no. 4, pp. 382–390, 2015.

[6] R. Gupta, P. Jain, and S. Vyas, “CFD Modeling and Simulation of 10KWE Biomass Downdraft Gasifier,” vol. 7, no. 4, pp. 0–2, 2017.

[7] ANSYS. ANSYS FLUENT 12.0 technical guide (12.0), 01 23; 2009