JMM: Journal of Mechanical and Manufacture
Vol. 3 No 1, October 2023 , pp. 1-11
DOI: 10.31949/jmm.v3i1.5682

e-ISSN: 2829-4424

FLUID DYNAMIC SIMULATION ON THE FLARE OF COMBUSTION OF GAS FROM BIOMASS GASIFICATION

Dian Susanto[1], Ari Wibowo [2] , Muhtar Kosim [3]

[1] Mechanical Engineering-University of Subang, [2]Mechanical Engineering-University of Subang,
[3]Architecture-University of Subang,
diansus4154@unsub.ac.id

Abstract: *The use of energy which always comes from fossil fuels will eventually run out, so the development of renewable energy or alternative energy is very important to maintain petroleum reserves and as a substitute for fossil fuels which are the main energy source. One alternative energy is biomass which has not been widely used by the gasification method. The gas produced by the gasification process is utilized by burning it in a flare to get a flame. In this study, the 3D simulation method with Computational Fluid Dynamics (CFD) was used to determine the temperature distribution on the flare walls using CFD simulations and to compare the temperature of the flare walls from the CFD simulation results with the test results. The results of this study, the distribution of combustion occurs in the flare with a temperature of 1106°C in the upper area close to the outlet boundary. The wall temperature comparison shows that the CFD simulation tends to be similar to the test results. This shows that computational fluid dynamic simulations can be used to predict fluid flow rates and combustion reactions.*

Keywords: *Biomass Gasification, CFD, Flare*

Submitted: 2023-06-15	Revised: 2023-10-01	Accepted: 2023-10-01

Introduction

Energy use today and for the future is enormous. If the energy used always comes from the use of fossil fuels, of course, over time it will run out. So the development of renewable energy or alternative energy is very important to maintain petroleum reserves and as a substitute for fossil fuels which are the main energy source. So it is necessary to look for alternative energy sources that can be renewed. One alternative energy source that is easy to find and which has not been widely used is biomass. One example of biomass is coconut shells in the utilization of coconut shells as an alternative energy producer using the gasification method (Sari, D. P., 2016).

Gasification is a process of converting compounds containing carbon to convert both liquid and solid materials into combustible gaseous fuels (CO, H2, CO2, CH4 and H2O) through a combustion process with limited air supply, namely between 20% and 40% stoichiometric air. The reactor where the gasification process takes place is called a gasifier (Ridwan, M., et al., 2018).

A one-way flow gasifier is a biomass gasification gasifier in which air (oxygen) flows simultaneously with the biomass fuel to the reactor. Air (oxygen) flows into the reactor due to low pressure (vacuum pressure). Low pressure on the reactor is carried out by means of suction. This suction process will not produce high pressure concentrations if the channel is blocked and will even stop the air flow so that a one-way flow gasifier will be safer than a driven airflow type gasifier. In addition, biomass can be supplied easily into the reactor because the biomass fuel simultaneously enters the air stream due to the fact that the reactor pressure is lower than atmospheric pressure. (Setioputro, N. T., et al., 2019).

The gas produced by the gasification process is burned in a flare to get a flame. So researchers carry out CFD simulations on flares to accurately predict fluid flow, heat transfer and combustion reactions.

Literature Review

This chapter will explain the introduction to the theory as a basis for research. Such as the understanding of biomass gasification, Flare and CFD (Computational Fluid Dynamics). Each of them is:

1. Gasification of Biomass

Gasification is the process of changing solid fuel in the form of biomass into gas by means of a thermochemical process by means of partial oxidation or a mixture of air-fuel at high temperatures. The results of combustion will be in the form of water vapor and carbon dioxide which is reduced so that it becomes a gas that produces energy (Suzaqi, A., et al. 2020).

Biomass is a solid fuel of biological origin that can be converted into renewable energy. Biomass combines solar energy and carbon dioxide into chemical energy in the form of carbohydrates through photosynthesis. The use of biomass as a fuel is a carbon neutral process because the carbon dioxide captured in the air during photosynthesis is released back into the air during the combustion process. (Setioputro, N. T., et al., 2019).

From this it can be concluded that in the gasification process, less oxygen is used than the air used for the combustion process. The gasification results consist of combustible gases, namely CO, H2, and CH4, besides that there are also non-combustible gases, such as CO2, H2O, fine dust, and tar..

a) Eco-Friendly Biomass Cycle

The biomass cycle requires several stages to obtain environmentally friendly biomass, while the several stages can be seen as shown in the image below:

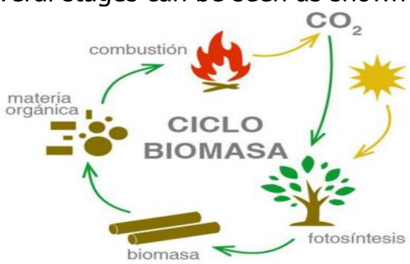

Figure 1. Geometry models

1) The process of photosynthesis in plants. Photosynthesis is a process experienced by plants which requires CO2 gas and sunlight where this process occurs during the day, however, this process occurs in plants or plants..
2) There are several biomass processes that fall into the category of biomass, namely; a) rice husks, b) coconut shells, c) corn cobs, d) straw, e) wood waste, f) leaves/branches, g) bagasse, h) firewood. From the above categories, all of them can become biomass. To become gasified biomass, a uniform size must be formed in order to get maximum gasification.
3) The process of organic material after it has been shaped into a uniform size, the biomass is dried in the sun to reduce the water content in the biomass to obtain a certain moisture value.
4) Combustion Process, this process is the burning of biomass, for example, like a bonfire, only some of the biomass becomes charcoal, the others become organic fine ash.
5) CO2 process, this process is obtained from the combustion of biomass which is then absorbed by plants to get good photosynthesis.

The material for this biomass is very easy to find in village areas and there are even some people who produce biomass that is supplied as a substitute fuel for Elfiji gas. This biomass cycle is continuous and long-term while there are still plants and humans.

b) Combustion and Gasification process

Biomass can be converted into synthesis gas that can be burned (combustible gas) as a substitute for gasoline through a thermochemical conversion process. The synthesis gas contains carbon dioxide (CO2) and hydrogen (H2) gases which are obtained from the combustion process followed by the gasification process. The thermochemical reaction process for the formation of synthesis gas is as follows:

1) Combustion process using a chemical formula, as follows.
   CHxOy (biomassa) + O2 (21% air) = CO2 + H2O + C + tar + calor
2) Meanwhile, in the gasification process, the chemical equation is: C + CO2 + Calor = 2CO, Carbon Dioxide + Charcoal + Heat = Carbon Monocide, H2O + C + Heat = H2 + CO, Water Vapor + Charcoal + Heat = Hydrogen + Carbon Monocide, Hydrogen + Carbon Monocide = Combustible Gas (Syngas).

The gasification reaction process can take place perfectly depending on the heat of reaction, in the gasification process several factors must be considered, starting from the reactor temperature in the range of 800 – 1000 °C. Moisture value in the biomass to be tested. Consumption of biomass in the reactor furnace is not too much and not too little. so that the heat generated by the combustion reaction process must be able to remain stable and continue to get the gasification reaction (Kumar, et al. 2009).

c) Scheme and Working Method of Biomass Gasification

This Biomass scheme is simple and easy to learn, the workings of Biomass Gasification can be seen in the cycle below, Biomass and air are burned in the reactor furnace then at the end of the cycle are sucked in by the blower before the combustion results produce gasification, the gas contained in the reactor enters the cyclone where the cyclone separates the syngas and particles (ash), after being separated the syngas goes to the cooling pipe where the syngas is cooled from 180 °C to 50 °C from this cooling produces Tar which is collected by the condensate.

After the syngas has cooled to the filter, the sucked tar will be filtered by the filter then the syngas enters the air pump (Blower) after that the air pump pushes the syngas to go to the flare, the syngas flare is burned to get the maximum flame, you have to pay attention to several factors starting from reactor temperature in the range of 800°C – 1000°C. Moisture value in the biomass to be tested. Consumption of biomass in the reactor furnace is not too much and not too little so that the heat generated by the combustion reaction process must be able to remain stable and continue to get the gasification reaction.

2. Flares

Flare gas is gas produced by the production and production or processing of oil or natural gas which is burned because it cannot be handled by the available production or processing facilities so it has not been utilized. From this definition, flare gas is an associated gas. While Flaring Gas is the combustion of Flare Gas in a stationary stack, both vertical and horizontal. The process of burning gas most often occurs at the top of the chimney by using a flame to burn the gas. The height of the flame depends on the volume of gas released, while the brightness and color depend on the composition of the gas (Upara, N., et al. 2016).

Flare is a safety system for a gas produced from processing or production by burning the gas. Apart from being a safeguard, burning gas flares aims to minimize environmental pollution because if the gas is discharged into the air without being burnt first, of course it will have a negative impact on the surrounding environment. Efficient combustion in a flame depends on achieving good mixing of the gaseous fuel and air (or steam). The height of the flame depends on the volume of gas released, while the brightness and color depend on the composition of the gas.

3. Computational Fluid Dynamics (CFD)

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of a system involving fluid flow, heat transfer and other physical processes (Pranatal, E., 2016). CFD is able to analyze and predict quickly and accurately. CFD can also be interpreted as a computer technology that allows users to study the dynamics of flowing objects or substances (Aprianto, D. D., et al. 2016).

The goal of CFD is to make accurate predictions about fluid flow, heat transfer, and chemical reactions in complex systems, involving one or all of these phenomena..

CFD is used by scientists and engineers to perform numerical experiments in a virtual laboratory. In terms of flow characteristics, CFD can show more detailed and accurate flow patterns that would be difficult and expensive, even impossible using experimental techniques. CFD has the ability to study systems under hazardous conditions at or after passing a critical point (including safety studies and accident scenarios). One example of an application that has been carried out is in the analysis of obtaining the forces and effects of a research that has a very long time and is in the deep sea.

Schowalter, et al 1989, stated that error tolerance in scaling can be reduced by the presence of CFD. In different research areas, the application of CFD is carried out as a comparison with experiments where it is possible to carry out experiments and being superior in cases where experiments are very difficult or even impossible to do.

Research Methodology

In the method used there are several stages where these stages can be seen in Figure 1.

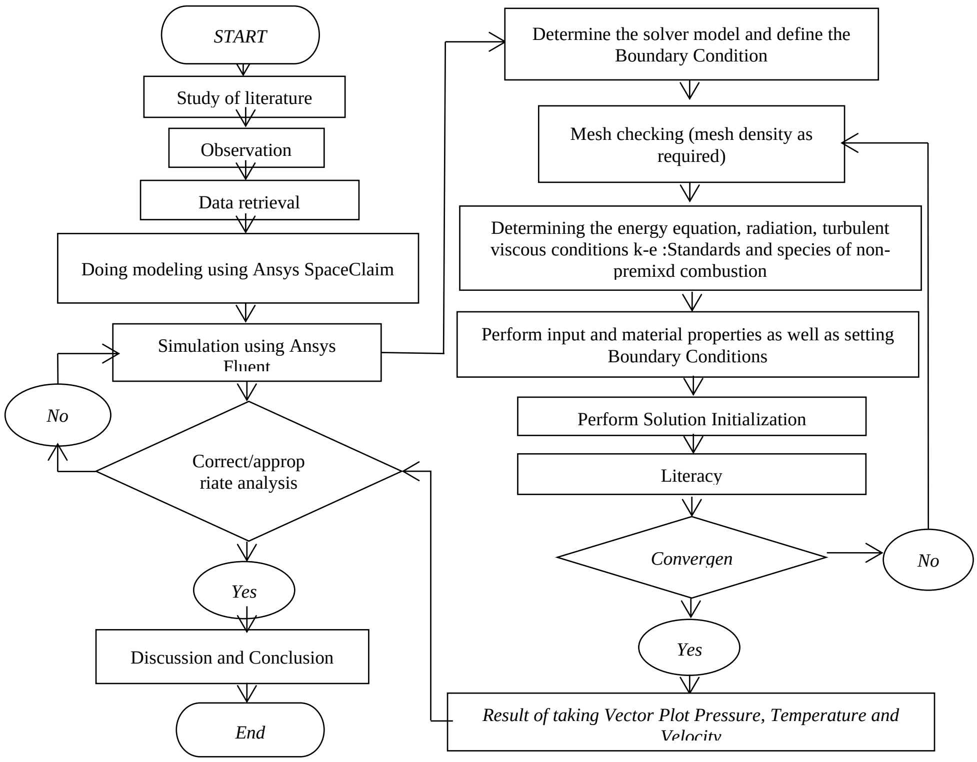

Figure 1. Flow Chart

1. Flowchart explanation

In accordance with the flow chart above, the explanation for each step can be explained as follows:

a) Study of literature

Literature study was conducted to find materials and theories related to this research and to make it easier to determine the process to be carried out during the research. Literature studies were obtained from journals and books.

b) Geometry Creation

The flare geometry is taken from the results of observations and data collection to facilitate the validation process. Ansys Fluent software is used to draw 3D geometric flare shapes.

c) Mesh Check

This step determines the success of the literacy process. When meshing has poor quality, the simulation process takes a long time and produces inaccurate data.

d) Determine the energy equation, radiation, turbulent viscous conditions k-e : Standard and non-premixed species

This section determines the energy equation and the Viscos turbulent k-epsilon condition with fluid property data and selection of non-premixed combustion species.

e) Process input and material properties and Boundary conditions

Entering material properties and Boundary conditions (boundary conditions) are entered with variables according to needs. These variables are obtained from literature studies in the form of pressure, speed and temperature.

f) Solution Initialization

The convergence criterion used is if the absolute residual value has reached 0.001 for all equations.

g) Contour and vector plots of pressure, temperature, and velocity

Contour plotting of temperature, pressure, velocity and turbulence along the flare is carried out to find out the phenomena that occur inside the flare.

2. Data collection procedures

The stages of data collection used in this study are as follows:

a) Measure flow rate using a pitot tube

Measure this flow rate using a pitot tube. The way this tool works is, install it on the flare pole, make sure it is straight and precise. Tie the part of the hose filled with water, then attach the pitot tube above the flare flow hose, after that the pitot tube is clamped to maintain its precision, as shown in Figure 2 below.

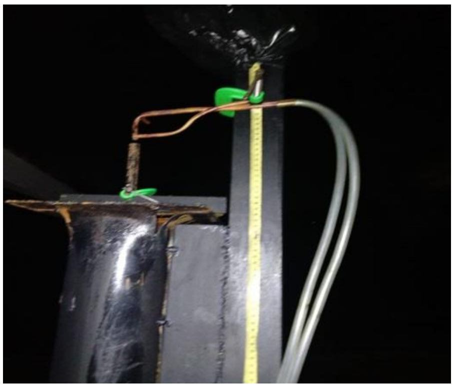

Figure 1 Mounting and calibrating the pitot tube

b) Measure the temperature of the gas entering the flare using a digital infrared thermometer

The way this thermometer works is that first make sure the tool is working and the battery is installed and suitable for use, then calibrate the tool and how to measure it, press the measure button on the tool then there is an infrared light pointing the red light at the flare material.

Figure 2 Flare temperature measurement using a thermometer

c) CFD simulation process using fluent ansys

In this study, researchers used the Ansys Student 2021 R1 software as shown in Figure 4 below.

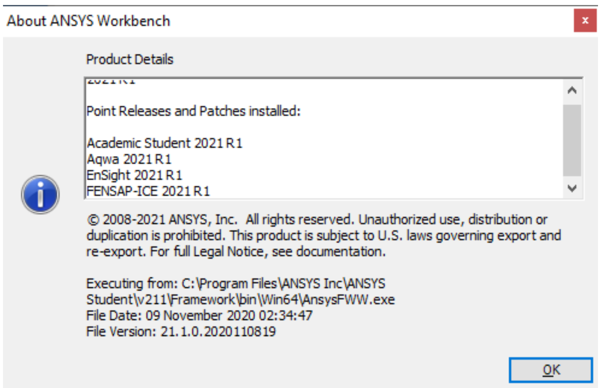

Figure 3. Versi ansys

In using the Ansys Student 2021 RI software, there are several steps and stages including; a) pre-processing, namely by making geometry, determining boundary conditions, and meshing. b) processing, At this stage there are many things that must be done in relation to determining the boundary conditions in a CFD simulation. This process is the most important process because almost all research parameters are processed in this stage such as models, materials, boundary conditions, solution methods, solution controls, solution initialization, calculation activities, and finally run calculations. c) Post-processing, the next step after carrying out the calculation process is to see the results of the calculation process. The calculation results can be interpreted into drawings, graphs, contours and fluid flow animations.

d) Convergence Criteria

Each equation that is run in the simulation has a residual that keeps changing and decreases in value. The smaller the residual, the more accurate the calculation will be. But in its application, the residual figure will continue to exist and continue to fluctuate. Therefore it is necessary to decide the right time to complete the calculation by determining the convergence criteria for each residual from the equations that are run.

In this simulation, the convergence criterion used for each residual is 0.001. These values are the right values because they can produce valid data.

Research Results and Discussion

After determining the research method, data collection, modeling and simulation using Ansync Fluent, next is the calculation and data analysis stage. And some calculation results and data analysis can be explained in the following sections.

1. Calculation of flow velocity on flares

   At this stage, some results are obtained by using the equation according to the values needed in the next process including:

   a) Flow rate

      The inlet and outlet flow rates obtained a value of v = 47,84 m/s.

   b) Flow discharge

      After obtaining a large cross-sectional area of 5.3 x 10^{-4} m^2, the amount of water discharge (Q) is: 2,53x10^{-2} m^3/s.

   c) Flare flow rate

      To calculate the value of the flow velocity at the flare at the air1, air2, and air3 inlets it can be calculated by equation (4). The constant values for this equation are obtained from the data. The value of the flow velocity at the water inlet 1 is 17,12368 m/s.

2. Pre-processing

   In the pre-processing stage there are several works including the creation of geometry, boundary conditions and meshing which are all done by the Ansys fluent program. The rest can be seen in the following explanation.

   a) Geometry

      Creating a geometry model can be done by utilizing the features found in Ansys Student 2021 R1. In this research, 3D geometry for this simulation uses Ansys SpaceClaim Geometry, as shown in Figure 5 below.

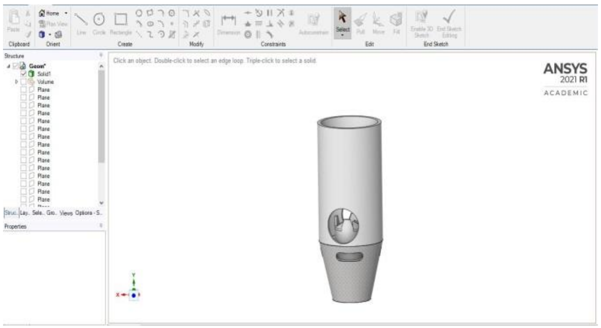

Figure 5. Geometry models

b) Meshing

The purpose of meshing is to form elements in component parts. This is to facilitate the pre-processing stage. Meshing in this study uses poly-hexcore.

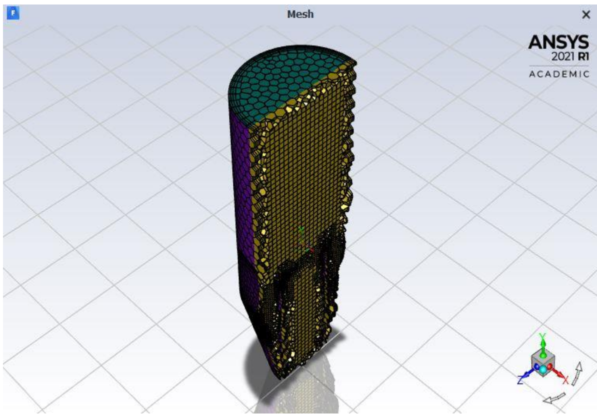

Figure 6. Geometry models

c) Boundary conditions

In this study there are 8 boundary conditions, namely air1a, air1b, air2a, air2b, air3a, air3b, fuel, and outlets as shown in Figure 7 below.

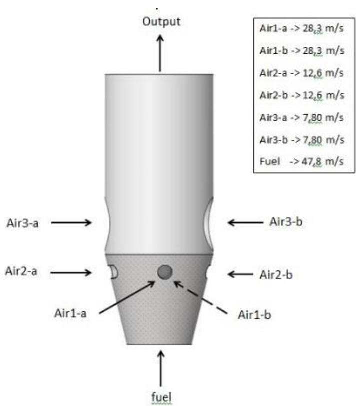

Figure 7. Geometry models

3. Post-Processing

After doing pre-processing, then in this process the calculation process is carried out. As for this process, there are several stages accompanied by the results, among others:

a) Speed contour display

The display of the CFD simulation results in the form of velocity contours is as shown in Figure 8 below:

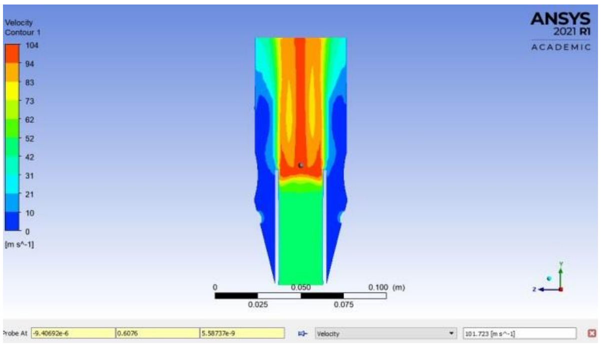

Figure 8. Speed Contour

The picture above shows the distribution of flow velocity in the flare pipe, there are different colors from the side of the wall to the center point of the pipe. It can be concluded that the lowest speed is on the blue side of the lower pipe wall and the highest is red at the pipe center point because there is a narrowing of the cross-sectional area resulting in zet flow, in the figure above shows an increase in speed from the fuel limit condition of 49.1 m/s to 101.7 m/s in the middle of the pipe there is a narrowing of the cross-sectional area. The distribution of fluid velocity is shown by the fluid flow in red, namely the highest speed with a value of 101.7 m/s and the low speed in the pipe, namely in blue with a value 5,28 m/s.

The narrowing of the cross-sectional area also produces turbulent flow so that the mixing of gas and air can be easily mixed properly and the flame continues to burn stably. As in the results of the velocity vector display below.

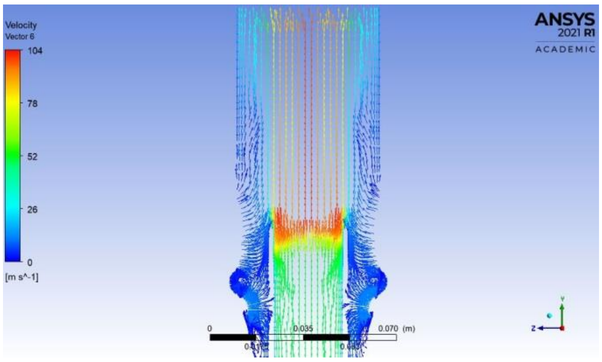

Figure 9. speed vector

b) Temperature contour display

The temperature calculation results are obtained from the CFD simulation results where the input temperature is assumed to be constant, namely 50°C at the fuel inlet. The magnitude of the value of the temperature distribution is depicted by color, for the lowest temperature the color is blue, followed by green, yellow and the highest is red as shown in Figure 10 below.

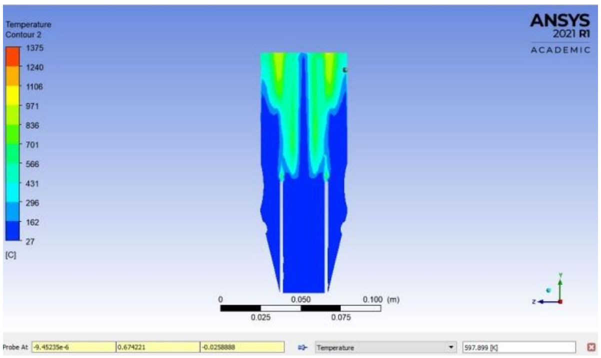

Figure 10. Temperature contour

In the figure above, it can be seen that the temperature contour changes color and the temperature value changes from blue to yellow. Where the distribution of combustion that occurs in the yellow area is 971°C to 1106°C, and the temperature distribution in light blue is 296°C to 431°C.

c) Wall temperature comparison

The following is the result of a comparison of the temperatures that occur in flares from the CFD simulation results, as shown in Figure 11 below:

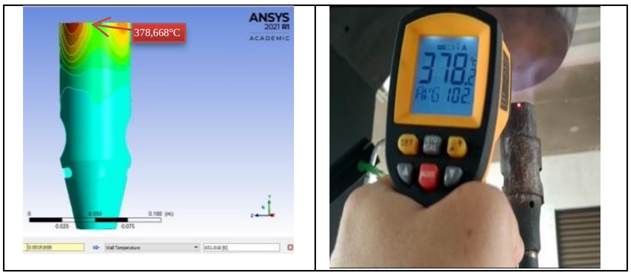

Figure 11. Wall temperature comparasion

The figure above shows the wall temperature in the outer area of the flare pipe, a comparison of the wall temperature from the CFD simulation results with the test results shows that there is a similarity in the value of the same temperature location on the flare.

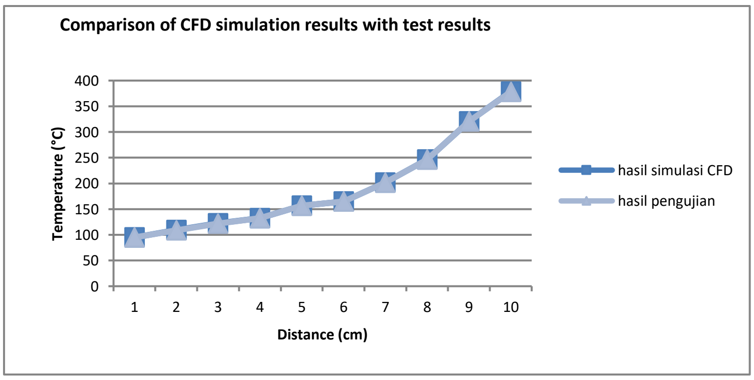

Figure 12. Comparison of simulation results with test values

From the comparison results in Figure 12, it shows that the CFD simulation has a tendency similar to the test results. This shows that computational fluid dynamic simulations can be used to predict fluid flow rates and combustion reactions.

Conclusion

In a computational fluid dynamics simulation study on gas combustion flares resulting from biomass gasification, it can be concluded that; a) the combustion distribution occurs inside the flare with a temperature of 1106°C in the upper area close to the outlet boundary, b) the wall temperature comparison results show that the CFD simulation has a tendency similar to the test results. This shows that CFD simulation can be used to predict fluid flow rates and combustion reactions.

Reference

Books

R. M. Bruce, F. Y. Donald, and H. O. Theodore, Mekanika Fluida, 1st ed. Jakarta: Gelora Aksara Pratama, 2004.

Articles in Journals/Magazines

Sari, D. P., et al. (2016). Pengaruh distribusi udara terhadap kinerja tungku gasifikasi sekam padi tipe downdraft continue. Surakarta : Teknik Mesin, UMS.

M. Ridwan, I. Indradjaja, N. Nugraha, and I. Taufik, "Pengujian Kinerja Dan Modifikasi Reaktor Downdraft Gasifikasi Biomassa 100 kW," J. Rekayasa Hijau, vol. 2, no. 2, pp. 157–164, 2018, doi : 10.26760/jrh.v2i2.2394.

Setioputro, N. T., et al. (2019). Kaji eksperimental gasifikasi biomassa reaktor tekanan vakum. MESA (teknik Mesin, elektro, sipil, arsitektur), Vol. 3, No. 2, 1-10.

Suzaqi, A., (2020). Pengaruh Laju Udara Dan Variasi Lubang Terhadap Biomassa The Influence Of Air Flow And Hole Variations Against The Flame Color Characteristics Of The Biomass Gasification Stove. Vol. 7, No. 3, 9286–9293.

Setioputro, N. T., et al. (2019). Perancangan Sistem Pendingin Pada Gasifikasi Biomassa Reaktor Aliran Searah Untuk Pengganti Bahan Bakar Bensin Mesin Generator 2 kVA. MESA (Teknik Mesin, Tek. Elektro, Tek. Sipil, Tek. Arsitektur). Vol. 3, No. 2, 11–21.

Kumar, A., et al (2009). Thermochemical biomass gasification: A review of the current status of the technology. Energies. Vol. 2, No. 3, 556–581, doi: https://doi.org/10.3390/en20300556.

Upara, N., et al. (2018). Studi eksperimen pemanfaatan gas suar (flare gas) menghasilkan daya listrik 15 kW. SEMRESTEK (Seminar rekayasa Teknologi), Submission 47, pp. 176-187.

Suhendi, E., et al. (2016). Pengaruh laju alir udara dan waktu proses gasifikasi terhadap gas producer limbah tangkai daun tembakau menggunakan gasifier tipe downdraft. JBAT (Jurnal Bahan Alam Terbarukan), Vol. 5, No. 2, doi : https://doi.org/10.15294/jbat.v5i2.6054

Alwan, H., (2019). Model gasifikasi biomassa menggunakan pendekatan kesetimbangan termodinamika stoikiometris dalam memprediksi gas produser. JIP (Jurnal Integrasi Proses), Vol. 8, No. 1, 31-38, doi : http://dx.doi.org/10.36055/jip.v8i1.5597.

Ilminnafik, N., (2017). Karakteristik api syngas pada gasifikasi downdraft dengan bahan biomassa sekam padi. ROTOR (Jurnal Ilmiah teknik mesin). Vol. 9, No. 1.

Pranatal, E., (2016). Numerical study darrieus savonius turbine for energy conversion at low speed marine current using CFD simulation. Surabaya: Magister Teknik, ITS Surabaya. Tesis 2016.

Aprianto, et al. (2014). Analisis aliran udara pada jembatan suramadu dengan menggunakan metode volume hingga. Kadikma (jurnal matematika dan pendidikan matematika), Vol. 5, No. 3, doi : https://doi.org/10.19184/kdma.v5i3.1380.

Atwood BT, Schowalter WR. Measurements of slip at the wall during flow of high-density polyethylene through a rectangular conduit. Rheol. Acta. 1989; 28:134–146.