ABSTRAK

Nowadays, the consumption of non-renewable energy is high. As a result, the availability of this energy is decreasing. Because of this, other energy alternatives are needed. One example of another option is the use of biomass. An example of using biomass is the gasification process. This process is carried out in a device called a gasification stove, namely a gasifier. A gasifier has been developed to produce an optimal combustion system. The form of development that can be done is to vary the dimensions and number of the air inlet. The constraints in this development are costs for manufacture and the potential for failure. Thus, this study offers an innovative simulation solution. The method used is Computational Fluid Dynamics (CFD). With CFD, the effect of dimensional variations and the number of air inlet can be modelled and computed to obtain the results without being experimental. The simulation results change the dimensions and quantity of inlet that affect each design's efficiency and temperature. The simulation results show that the highest efficiency of downdraft gasifier, updraft gasifier, and entrained flow gasifier was A1, C1, and B3, which value 11,393%, 9,354%, and 50,885%, respectively. Besides, the temperature of A1, C1, and B3, were 1055,25 K, 1383,44 K, and 3580,15 K, respectively. Therefore, this study suggested that a gasifier's development could be optimized first through simulation to reduce costs and failures.

Keywords: air inlet, combustion, computational fluid dynamic, dimensional, efficiency, gasification.