INTISARI

Penelitian ini bertujuan untuk merancang *boiler Circulating Fluidized Bed* (CFB) dengan biomassa sebagai bahan bakar utama. Biomassa adalah sumber energi yang ramah lingkungan, sehingga dapat mendukung upaya mengurangi pencemaran lingkungan. Penelitian ini diharapkan dapat memberikan kontribusi positif terhadap implementasi kebijakan energi terbarukan di Indonesia, sejalan dengan visi untuk mencapai ketahanan energi yang berkelanjutan.

Perancangan melibatkan perhitungan dimensi *boiler* dan simulasi *Computational Fluid Dynamics* (CFD). Perhitungan didasari oleh target kapasitas termal dan properti kandungan biomassa yang digunakan sebagai bahan bakar. Simulasi dilakukan untuk memahami distribusi suhu serta aliran partikel fluida di seluruh *boiler* selama proses pembakaran.

Hasil penelitian menunjukkan bahwa *boiler* yang dirancang memiliki tinggi dimensi *furnace* sebesar 29,4 m dengan 2 *cyclone* berdiameter 6,55 m. Hasil simulasi numerik ditampilkan dalam bentuk visualisasi aliran dan distribusi suhu di dalam *boiler*. Simulasi tersebut menunjukkan suhu sebesar di daerah *furnace* sebesar 600 – 700 °C dan kecepatan 0 – 20 m/detik. Hasil simulasi juga ditunjukkan dalam visualisasi abrasi yang mungkin terjadi di dalam *boiler*, tepatnya di daerah *furnace* bawah dan dinding *cyclone*.

Kata kunci: Kecepatan, suhu, *furnace*, *cyclone*, visualisasi, abrasi

ABSTRACT

Due to the extensive use of coal as a fuel for power generation, this study aims to design a Circulating Fluidized Bed (CFB) boiler using biomass as the primary fuel source. Biomass is an environmentally friendly energy source and thus supports efforts to reduce environmental pollution. This research is expected to make a positive contribution to the implementation of renewable energy policies in Indonesia, in line with the vision of achieving sustainable energy resilience.

The design process involves calculating the boiler dimensions and conducting Computational Fluid Dynamics (CFD) simulations. The calculations are based on the targeted thermal capacity and the properties of the biomass used as fuel. The simulations are performed to understand the temperature distribution and fluid particle flow throughout the boiler during the combustion process.

The results show that the designed boiler has a furnace height of 29.4 meters with two cyclones, each with a diameter of 6,55 meters. The numerical simulation results are presented in the form of flow and temperature distribution visualizations inside the boiler. The simulations indicate a furnace temperature ranging from 600 to 700 °C and fluid velocity between 0 to 20 m/s. The simulation also visualizes potential abrasion areas within the boiler, specifically in the lower furnace region and along the cyclone walls.

Keywords: *Velocity, temperature, furnace, cyclone, visualization, abrasion*