Analisis pemodelan variasi posisi inlet cyclone gas burner dengan metode simulasi CFD = Modelling analysis of inlet position variation on cyclone gas burner with CFD simulation method

Choky Jonathan, author

Abstrak

Penelitian ini memperlihatkan fenomena aliran fluida campuran antara udara dan gas sintetik di dalam cyclone gas burner. Pengamatan berfokus pada fenomena intensitas turbulen, energi kinetik turbulen, dan kecepatan aliran percampuran udara dan gas sintetik akibat variasi posisi inlet cyclone gas burner dengan pemodelan menggunakan ANSYS Fluent. Pengamatan pemodelan fenomena percampuran ini digunakan untuk melihat desain yang paling optimum untuk cyclone gas burner dengan debit konstan aliran udara dan gas sintetik masing-masing 11,38 x 10-4 m3/s dan 8,06 x 10-4 m3/s. Gas sintetik merupakan produk gasifikasi biomassa sekam padi tipe fixed bed downdraft gasifier dengan komposisinya 50% N2, 3% CH4, 18% H2, 19% CO, dan 10% CO2.

<hr>

This research showed the phenomena of fluid flow between the mixing flow between air and synthetic gas in cyclone gas burner. The observations focused on the phenomenon of turbulence intensity, turbulent kinetic energy, and the flow velocity of air and synthetic gas mixture as a result of variations in cyclone gas burner inlet position using ANSYS FLUENT modeling. Observating of mixing phenomena modeling is used to view the most optimum design for cyclone gas burner with constant air and synthetic gas each flow rates is 11,38 x 10-4 m3/s and 5,5 x 10-4 m3/s. Synthetic gas is rice husk biomass gasification product using fixed bed downdraft gasifier with 50% N2, 3% CH4, 18% H2, 19% CO, and 10% CO2 composition.