Cfd simulation of turbulent flows in proto x-3 bioenergy micro gas turbine combustor using std k-? and rng k-? model for green building application

Asyari Daryus, author

Abstrak

This paper presents a numerical analysis of gas flow in the annular combustion chamber of a Proto X-3 Bioenergy micro gas turbine for green building applications. The computational fluids dynamics (CFD) simulation was conducted in two dimensions, turbulent flow and gas phase combustion, with the goal of comparing the effects of different models in real conditions. Two different turbulence models, standard (STD) k-? and renormalization group (RNG) k-?, were applied for simulations. The fuel used was biogas produced from animal waste. Fuel consumption was assumed to be 100 kJ/s for simulations. The results of the simulations were analyzed and compared for reference. The temperature and the mass fraction of CH4, H2, O2, and CO2 distributions gave almost the same results for both models; therefore, both models (STD k-? and RNG k-?) could be used to represent the combustion process phenomenon without many significant differences.