Paper Number FBC2005 - 78013

COMBUSTION MODELING OF MUNICIPAL WASTE COMBUSTOR

Ahmad Hussain[1,*], Farid Nasir Ani[1], Amer Nordin Darus[1], Mohammed Fadzil Adnan[2], Adi Surjosatyo[3], Azeman Mustafa[4], Arshad A. Salema[4]

[1]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, Skudai 81310, Johor, Malaysia
[2]Enviornmental and Bioprocess Technology Centre, SIRIM Berhad, 40911, Shah Alam, Malaysia
[3]Mechanical Engineering Department, University of Indonesia, Kampus Baru, Depok, 16424, Indonesia
[4]Faculty of Chemical and Natural Resources Engineering, Universiti Teknologi Malaysia, Skudai 81310, Johor, Malaysia
*Corresponding Author: Phone: +607-5534785, Fax: +607-5566159
*Email: ahmad@siswa.utm.my

ABSTRACT

The average amount of municipal solid waste (MSW) generated in Malaysia is 0.5–0.8 kg/person/day and has increased to 1.7 kg/person/day in major cities. Due to rapid development and lack of space for new landfills, big cities in Malaysia are now switching to incineration. However, a major public concern over this technology also is the perception of the emission of pollutants of any form. Design requirements of high-performance incinerators are sometimes summarized as the achievement of 3Ts (time, temperature, and turbulence). An adequate retention time in hot environment is crucial to destroy the products of incomplete combustion and organic pollutants. Also turbulent mixing enhances uniform distributions of temperature and oxygen availability. CFD modeling is now in the development phase of becoming a useful tool for 3D modeling of the complex geometry and flow conditions in incinerators. However, CFD flow simulations can enable detailed parametric variations of design variables. CFD modeling of an industrial scale MSW incinerator was done using FLUENT. The 3D modeling was based on conversation equations for mass, momentum and energy. The differential equations were discretized by the Finite Volume Method and were solved by the SIMPLE algorithm. The k-ε turbulence model was employed. The meshing was done using Gambit 2.0. The cold flow simulations were performed initially to develop the flow and velocity field. Numerical simulations of the flow field inside the primary and secondary combustion chambers provided the temperature profiles and the concentration data at the nodal points of computational grids. Parametric study was also done to minimize the NO_x emissions.

Keywords: CFD, combustion, flow simulations, MSW, turbulent mixing, temperature profile