

Computational Fluid Dynamics (CFD) Simulation of Multiphase Flow in Biogas Digester

V. S. Kshirsagar¹, P. M. Pawar¹

¹Department of Civil Engineering, SVERI College of Engineering, Pandharpur, Maharashtra, India

Abstract

Effective suspension and settling are critical for controlling biomass retention in a biogas digester. This paper developed a computational fluid dynamics (CFD) model to simulate the hydrodynamic characteristics of multiphase flow in biogas digester. This is carried out by using COMSOL Multiphysics® software for understanding the behavior of slurries of different viscosity. This study helps to understand to take the judgment of water to solid waste ratio, for the improvement of gas production in the digester.