WATER RESEARCH 45 (2011) 2082–2094

Available at www.sciencedirect.com

ScienceDirect

journal homepage: www.elsevier.com/locate/watres

CFD investigation of turbulence models for mechanical agitation of non-Newtonian fluids in anaerobic digesters

Binxin Wu

Philadelphia Mixing Solutions Ltd., Palmyra, PA 17078, USA

ARTICLE INFO

Article history:
Received 19 July 2010
Received in revised form
15 December 2010
Accepted 20 December 2010
Available online 24 December 2010

Keywords:
Anaerobic digester
Computational fluid dynamics
Turbulence model
Non-Newtonian fluid
Mechanical mixing

ABSTRACT

This study evaluates six turbulence models for mechanical agitation of non-Newtonian fluids in a lab-scale anaerobic digestion tank with a pitched blade turbine (PBT) impeller. The models studied are: (1) the standard $k-\varepsilon$ model, (2) the RNG $k-\varepsilon$ model, (3) the realizable $k-\varepsilon$ model, (4) the standard $k-\omega$ model, (5) the SST $k-\omega$ model, and (6) the Reynolds stress model. Through comparing power and flow numbers for the PBT impeller obtained from computational fluid dynamics (CFD) with those from the lab specifications, the realizable $k-\varepsilon$ and the standard $k-\omega$ models are found to be more appropriate than the other turbulence models. An alternative method to calculate the Reynolds number for the moving zone that characterizes the impeller rotation is proposed to judge the flow regime. To check the effect of the model setup on the predictive accuracy, both discretization scheme and numerical approach are investigated. The model validation is conducted by comparing the simulated velocities with experimental data in a lab-scale digester from literature. Moreover, CFD simulation of mixing in a full-scale digester with two side-entry impellers is performed to optimize the installation.

1. Introduction

Anaerobically digesting organic waste is an economical solution to the pressing concerns of the environment and utilizing sustainable energy. Mixing is an important operation that homogenizes anaerobic bacteria, nutrients, and temperature throughout the digester to maximize biogas production. The common mixing methods involve the use of gas mixers, mechanical stirring, and mechanical pumping, among which the mechanical stirring has proven to be the most efficient method in terms of mixing intensity per unit power consumption (Wu, 2009, 2010b). With the exception of highly viscous fluids mixed at a low impeller speed, mixing in the digesters always creates turbulence. When using computational fluid dynamics (CFD), choosing an appropriate turbulence model is critical to characterize the flow fields. Generally, the approaches involved in modeling turbulence are direct numerical simulation (DNS), large eddy simulation (LES), and the eddy viscosity models. Both the DNS and LES are too computationally expensive for most engineering applications despite the fact that the DNS provides the best solution to turbulent flow and the LES shows a high accuracy in capturing large-scale chaotic structures. By contrast, an economic approach is to solve an eddy viscosity model that is based on the Reynolds-averaged Navier–Stokes (RANS) equations with a turbulence closure.

Among a large family of turbulence closures (zero-, one-, and two-equation, etc.), the standard $k-\varepsilon$ model has been the most popular one used to simulate mixing (Sahu et al., 1999; Alexopoulos et al., 2002; Chapple et al., 2002; Pruvost et al., 2004; Kukukova et al., 2005; Mostek et al., 2005; Deglon and Meyer, 2006; Vakili and Nasr Esfahany, 2009). Sahu et al. (1999) introduced a zonal modeling method to predict mixing by five different axial-flow impellers in a tank. They claimed that predictions of the turbulent kinetic energy (k) closely match the laser Doppler anemometry (LDA) measurements, and proposed a new method to estimate the turbulent energy

E-mail address: bwu@philamixers.com.
0043-1354/$ – see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.watres.2010.12.020

Nomenclature

B	baffle width, m
C	impeller clearance, m
CFD	computational fluid dynamics
CPU	central processing unit
d	diameter of moving zone, m
D	impeller diameter, m
DNS	direct numerical simulation
$\vec{F}$	body force, N
G	generation for k or ϵ
H	liquid height, m
k	consistency coefficient, Pa s^n
k	turbulent kinetic energy, m^2/s^2
K_s	empirical data
LDA	laser Doppler anemometry
LES	large eddy simulation
MRF	multiple reference frame
n	power-law index
N	rotating speed, rpm
N_P	power number, dimensionless
N_Q	flow number, dimensionless
p	pressure, Pa
P	power, hp or kW
PBT	pitched blade turbine
Q	flow rate, m^3/s
RANS	Reynolds-averaged Navier–Stokes
Re	Reynolds number, dimensionless
Re_g	generalized Reynolds number
RSM	Reynolds stress model
RT	Rushton turbine
S	source term for k or ϵ
t	time, s
T	tank diameter, m
TS	total solids concentration, g/l
$\vec{u}$	absolute velocity, m/s
U_∞	average velocity, m/s
$\bar{v}$	average velocity, m/s
Y	dissipation for k or ϵ

Greek symbols

β	left (or right) inclined angle, deg
γ	up (or down) inclined angle, deg
Γ	effective diffusivity
$\dot{\gamma}$	shear rate, s^{-1}
δ	error, dimensionless
ϵ	dissipation rate of turbulent kinetic energy, m^2/s^3
η	non-Newtonian viscosity, Pa s
θ	spacing between two impellers, deg
ρ	density, kg/m^3
$\bar{\bar{\tau}}$	viscous stress, N/m^2
ω	specific dissipation rate, 1/s

dissipation rate (ϵ). Alexopoulos et al. (2002) developed a two-compartment model to simulate the turbulent flow in a pilot plant reactor by varying vessel size, impeller diameter, agitation rate, and viscosity. The model validation was conducted in a non-homogeneous liquid–liquid dispersion process, and an excellent agreement was obtained between predicted and measured values on the droplet size distributions over a wide range of experimental conditions. Chapple et al. (2002) reported that the power number is independent of the blade thickness and stays constant for $Re > 2 \times 10^4$ while mixing with a pitched blade turbine (PBT) impeller via the LDA validation. Pruvost et al. (2004) assessed the standard k–ω model for a marine impeller in a torus reactor by comparing the CFD predictions with the LDA data. Kukukova et al. (2005) and Mostek et al. (2005) simulated the flow fields and homogenization in cylindrical vessels with multiple impellers on a central shaft to check the velocity profiles, power and pumping numbers, in which the PBT and standard Rushton turbine (RT) impellers were used. The simulated results were shown to closely agree with the experiments from the literature. Deglon and Meyer (2006) numerically investigated mixing by a RT impeller in a 15 cm diameter tank. Their studies showed that the standard k–ε model solved with the multiple reference frame (MRF) method can accurately predict the turbulent kinetic energy, provided very fine grids (nearly 2 million control volumes for half of the tank) are coupled with a higher-order discretization scheme. However, the results indicated that the flow field and mean fluid velocity predictions are not strongly influenced by either the grid resolution or the discretization scheme. Vakili and Nasr Esfahany (2009) studied the effects of agitator speed, impeller diameter, baffle width and impeller clearance on turbulent flow field in the tank with a two-blade impeller and four baffles. The model calculations were validated against the specifications in an unbaffled tank reported by Alexopoulos et al. (2002).

Despite the wide and intense utilization of the standard k–ε model, the model has deficiencies such as poorly simulating non-equilibrium boundary layers. Thus, examination of the other turbulence models remains an active topic of CFD research (Jaworski et al., 1998; Jaworski and Zakrzewska, 2002; Aubin et al., 2004; Murthy and Joshi, 2008). Jaworski et al. (1998) applied the RNG (renormalization group) k–ε model to simulate mixing by a hydrofoil impeller in a cylindrical tank, and obtained a good agreement of numerical predictions with the LDA velocity data. Later, Jaworski and Zakrzewska (2002) checked six turbulence models involving the standard k–ε, the RNG k–ε, the realizable k–ε, the Chen–Kim k–ε, the optimized Chen–Kim k–ε, and the Reynolds stress model (RSM) in a flat-bottomed tank with a PBT impeller and four baffles. They compared the simulated tangential and axial mean velocity components as well as the turbulence kinetic energy with LDA data for the wall jet region in the tank, and concluded that (1) the tangential velocity was irrespective of the turbulence model, (2) the axial velocity was well predicted using the standard k–ε and the optimized Chen–Kim k–ε models, and (3) the turbulent kinetic energy was significantly under-predicted by all the turbulence models. Aubin et al. (2004) studied the effects of the standard k–ε and the RNG k–ε models on the numerical solution in a tank stirred by a PBT impeller, and showed that these two models under-predict the k value in the discharge jet of the impeller through the comparison of simulated and LDA results. Murthy and