Bioresource Technology 138 (2013) 297–306

Contents lists available at SciVerse ScienceDirect

Bioresource Technology

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

Effects of turbulence modelling on prediction of flow characteristics in a bench-scale anaerobic gas-lift digester

A.R. Coughtrie *, D.J. Borman, P.A. Sleigh

School of Civil Engineering, University of Leeds, Leeds LS2 9JT, UK

HIGHLIGHTS

- Flow in a gas-lift digester is investigated using computational fluid dynamics.
- The effect of four RANS turbulence models on the flow characteristics is shown.
- The Transition SST turbulence model is determined to be most accurate for this case.
- ANSYS Fluent and OpenFOAM are used to show solver independence.
- The accuracy of a singlephase approximation is examined using a multiphase model.

ARTICLE INFO

Article history:
Received 21 December 2012
Received in revised form 21 March 2013
Accepted 24 March 2013
Available online 6 April 2013

Keywords:
Anaerobic digestion
CFD
Turbulence modeling
Gas-lift
Multiphase

ABSTRACT

Flow in a gas-lift digester with a central draft-tube was investigated using computational fluid dynamics (CFD) and different turbulence closure models. The k-ω Shear-Stress-Transport (SST), Renormalization-Group (RNG) k-ϵ, Linear Reynolds-Stress-Model (RSM) and Transition-SST models were tested for a gas-lift loop reactor under Newtonian flow conditions validated against published experimental work. The results identify that flow predictions within the reactor (where flow is transitional) are particularly sensitive to the turbulence model implemented; the Transition-SST model was found to be the most robust for capturing mixing behaviour and predicting separation reliably. Therefore, Transition-SST is recommended over k-ϵ models for use in comparable mixing problems. A comparison of results obtained using multiphase Euler–Lagrange and singlephase approaches are presented. The results support the validity of the singlephase modelling assumptions in obtaining reliable predictions of the reactor flow. Solver independence of results was verified by comparing two independent finite-volume solvers (Fluent-13.0sp2 and OpenFOAM-2.0.1).

© 2013 Elsevier Ltd. All rights reserved.

1. Introduction

The desire to extract the embodied energy from within what are currently waste products has driven the increased use of anaerobic biogas digesters. As a result the stability and efficiency of the digesters has become of greater concern. The process of anaerobic digestion turns organic wastes into methane, carbon dioxide (biogases) and an organic waste product of reduced volume, with a lower pathogen load than the original material. The biogas produced from a fully operational stable digester is expected to be approximately 65% methane and 35% carbon dioxide by volume, this gas can then be used as fuel to heat the digester and other parts of the biogas plant or in the generation of electricity (Taricska et al., 2009). A number of different factors affect the stability of anaerobic digesters (AD's) including the temperature, substrate content and mixing of the slurry during digestion. For example, how well mixed the slurry is will affect the pH distribution throughout the digester; methane producing bacteria are highly sensitive to pH and even small variations can have a substantial effect. Mixing is also useful in preventing settling of suspended biomass and the build-up of a scum layer on the slurry surface which can inhibit the escape of the biogas. As such, a well-mixed homogenous slurry is necessary for stable, controlled anaerobic digestion (Turovskiy and Mathai, 2006).

Due to the nature of the slurries used in the digesters and the size of full scale industrial plants, experimental methods of determining the flow characteristics are expensive and complicated. Computational fluid dynamics (CFD) provides an excellent method of assessing the flow characteristics and mixing effectiveness under different digester configurations without the time and expense of experimental studies. Over the past 20 years, research work describing numerical modelling of anaerobic digesters has been undertaken widely; with CFD being used to assess the mixing in

* Corresponding author. Tel.: +44 1133433292.
E-mail address: A.R.Coughtrie@leeds.ac.uk (A.R. Coughtrie).

0960-8524/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.biortech.2013.03.162

anaerobic digesters of different types. This includes assessment and development of CFD procedures for use with mechanically mixed digesters (Wu, 2010a; Joshi et al., 2011; Bridgeman, 2012). Modelling of mechanically mixed digesters has shown that the type of impeller and flow direction effects the mixing efficiency, with up mixing being found to be more efficient than down (Wu, 2010b; Aubin et al., 2004). Yu et al. (2011) also investigated mechanically mixed AD's and showed the potential of helical ribbon impellers in the mixing of high solids digesters and provided insight into the minimum power requirements. Additionally high solids AD's typically contain slurries of a non-Newtonian nature which have been shown to produce significantly different flow patterns to Newtonian fluids when modelled (Wu and Chen, 2008). Numerical modelling has also been used to investigate flow and mixing in gas lift digesters, using tracers in full scale AD's to monitor mixing time and showing that for internal loop gas lift AD's transient oscillatory behaviour can sometimes be found (Terashima et al., 2009). Oey et al. (2003) showed that CFD modelling can be used to predict flow patterns in gas lift AD's. Mudde and Van Den Akker (2001) described how such modelling can be used to design and tune gas lift AD's and Karim et al. (2007) used CFD to alter the flow characteristics and reduce the stagnation region, by modifying the geometry of a bench scale anaerobic gas lift digester. There has however been no definitive methodology produced defining the most appropriate models and approach to use in predicting the complex flow in anaerobic digesters. One of the significant factors is that slurry being mixed in many bioreactors, including bench scale reactors from where experimental data is often obtained, has Reynolds numbers indicating flow to be in the transitional turbulent region. This type of flow is known to be difficult to model and many common turbulence models fail to correctly resolve the flow field. This is compounded by the non-Newtonian nature of many slurries which can significantly alter Reynolds numbers throughout the digester where internal shear stresses vary. Published literature has not fully addressed the issue of which turbulence models are appropriate, nor what criteria should be adopted in selecting one for slurries of particular rheology. Failure to simulate turbulence correctly in non-Newtonian, transitional flow regimes may result in an inability to capture the important flow characteristics responsible for mixing reliably. There have been a small number of studies into the effects of turbulence modelling on the CFD results for anaerobic digesters (Wu, 2010b, 2011; Joshi et al., 2011; Bridgeman, 2012). The majority of CFD modelling of anaerobic digesters tend to rely on the standard k-ϵ turbulence model with wall functions (Vesvikar and Al-Dahhan, 2005; Meroney, 2009; Mudde and Van Den Akker, 2001; Oey et al., 2003). Often little justification for this choice is given and may be due to it being a good general purpose turbulence model which has been found suitable for a wide range of flows. This is not however the case where transitional flows occur, a factor which has been overlooked in previous studies. This approach impacts on the reliability of solutions as there is potential for significant variability in predictions for key phenomena, such as separation points, and thus stagnation zone size. Reduced accuracy in solutions may result, with the k-ϵ being shown to delay or fail in predicting wall separation resulting from adverse pressure gradients (Menter, 2011). As such, the first part of this study was focused on determining the factors affecting the choice of turbulence model in gas recirculation digesters; particularly in regard to low Reynolds number (*Re*) flow, transitional flows and boundary layer separation. Additionally, a comparison was made between results for two alternative, finite volume based, CFD solvers (ANSYS Fluent 13.0sp2 and OpenFOAM 2.0.1) in order to assess the solver independence of the predictions.

There are a number of options available when simplifying the multiphase gas driven digester problem for CFD to reduce the computational expense. Karim et al. (2007) used an empirical approximation for the flow at the top and bottom of the draft tube of their digester reducing the model to a singlephase problem by neglecting the flow in the draft tube. This assumes that the gas hold up (i.e. the dispersed gas volume fraction (Sieblist and Lübbert, 2010)) is not significant in the main annular section of the digester (see Fig. 2.1), allowing for the gas-phase to be neglected and an empirical fluid velocity formulation applied at the top of the draft

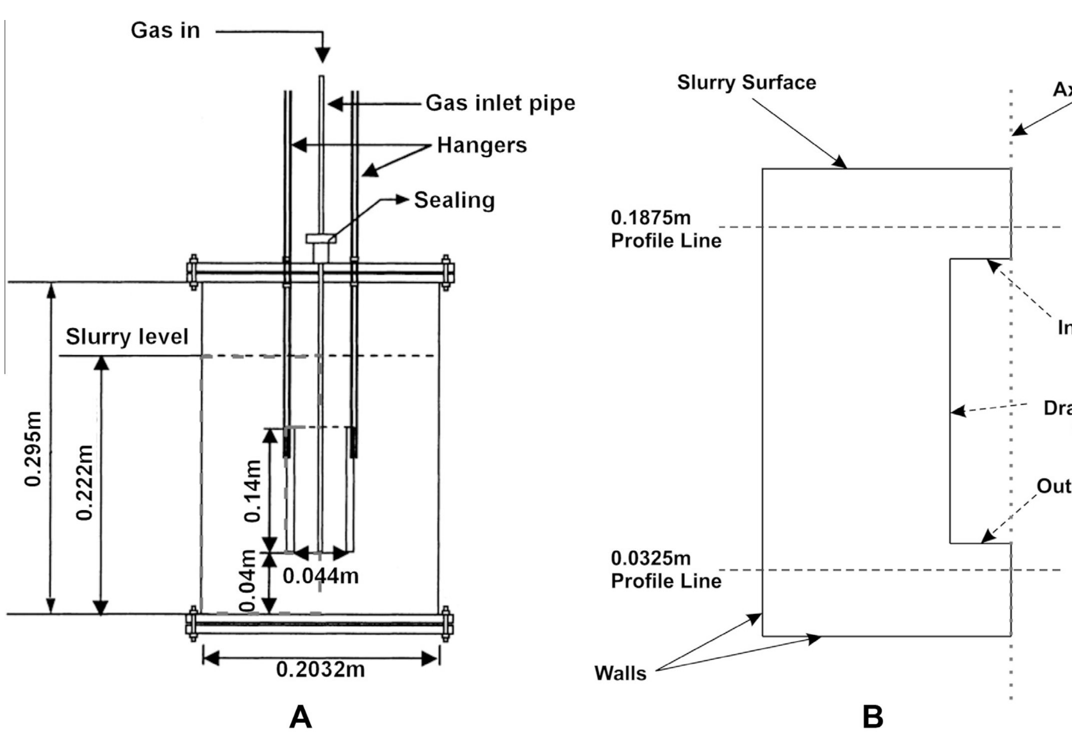

Figure 2.1. Bench scale digester geometry (Karim et al., 2004) (A) and the computational geometry for the singlephase model (B).

Download English Version:

https://daneshyari.com/en/article/7082453

Download Persian Version:

https://daneshyari.com/article/7082453

Daneshyari.com