Chemical Engineering Science xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Predicting flow and residence time in alumina digestion vessels

David S. Whyte [a], Gary J. Brown [a], David F. Fletcher [b,*]

[a] Alcoa of Australia Limited, Kwinana, Western Australia 6966, Australia
[b] School of Chemical and Biomolecular Engineering, University of Sydney, Sydney 2006, New South Wales, Australia

ARTICLE INFO

Article history:
Received 28 April 2016
Received in revised form 28 November 2016
Accepted 15 December 2016
Available online xxxx

Keywords:
Residence time distribution
Turbulence modelling
Scalar transport
Particle tracking
Digestion vessel

ABSTRACT

Understanding the flow and residence time behaviour in reaction vessels is of crucial importance to improve product yield or the extraction of mineral species from ores. Digestion vessels, used in the Bayer process, are an example of the latter. Transient Computational Fluid Dynamics (CFD) models were used to investigate the impacts of a variety of turbulence models, along with the effects of mesh refinement on the performance of two different vessel geometries. The main focus of the research work was the efficacy of a variety of turbulence models: two-equation (k-ε, SST), Reynolds Stress (SSG RSM) and Scale Adaptive Simulation (SAS-SST). The predicted velocity profiles are compared with high quality Laser Doppler Velocimetry (LDV) data. Residence times are calculated via a convected scalar (tracer) and Lagrangian tracking of particles. The calculated residence times are compared with salt tracer estimates from experiments. Generally, there was good agreement between experimental data and results obtained from tracing a passive scalar or tracking neutrally-buoyant particles, with the three results being closer when the SAS-SST model was used compared with the k-ε model. In the case of the strongly swirling flow caused by the tangential entry, the tracer was in closer agreement with the experimental data but the Lagrangian tracking results improved considerably with the use of the SAS-SST model. The importance of small-scale turbulence structures in determining the flow profile and residence times of the vessels is highlighted in this study.

1. Introduction

Digestion plays a fundamental part in the Bayer process for alumina production, with the objective of this processing stage being to extract the available alumina from the bauxite by dissolving it in a hot caustic soda solution (Gerard and Stroup, 1963). Digester vessels typically have a height to diameter ratio of close to 3:1. Some vessels use mechanical agitation, but those examined in this study have a single inlet stream at the top of the vessel and a single, centrally-located, outlet at the bottom of the vessel, as in the schematic shown in Fig. 1. The flow behaviour in these vessels is therefore dominated by the design and positioning of the inlet nozzle. An understanding of the flow behaviour in these vessels is important, as adequate residence time and mixing are critical to achieving sufficient extraction of alumina from bauxite particles of varying diameter.

Past CFD modelling of digestion vessels within the organisation used the standard k-ε model in steady-state simulations (Brown and Fletcher, 2005). While the simulations of top-fed digesters were considered to give acceptable agreement with experimental data, the same could not be said for the simulations for tangentially-fed vessels. At the time of that work, scale-resolving turbulence models were at their development stage and computing resources were vastly more expensive than they are now.

Woloshyn et al. (2006) used CFD to investigate a range of digester feed configurations. Their study considered full-scale digesters using the k-ε model and the Reynolds Stress Model (RSM) for turbulence. Residence Time Distributions (RTD) were determined for individual particles of various sizes. However, no validation of the simulations was conducted.

Brown et al. (2014) studied the flow in precipitation vessels using CFD. These vessels have flow rates and velocity scales which are comparable with those for the digesters considered here. The work evaluated a variety of turbulence models and validated the numerical predictions against experimental measurements. The authors found that using a turbulence model that was able to capture the unsteady, large-scale turbulence structures (SAS-SST) gave results that matched the experimental data very well.

Thus, given improved modelling techniques and computational resources, it is now possible to examine the performance of more sophisticated turbulence modelling approaches. Below, we give a brief description of the experimental study used for validation

---

* Corresponding author.
E-mail address: david.fletcher@sydney.edu.au (D.F. Fletcher).

http://dx.doi.org/10.1016/j.ces.2016.12.036
0009-2509/© 2016 Elsevier Ltd. All rights reserved.

Please cite this article in press as: Whyte, D.S., et al. Predicting flow and residence time in alumina digestion vessels. Chem. Eng. Sci. (2016), http://dx.doi.org/10.1016/j.ces.2016.12.036

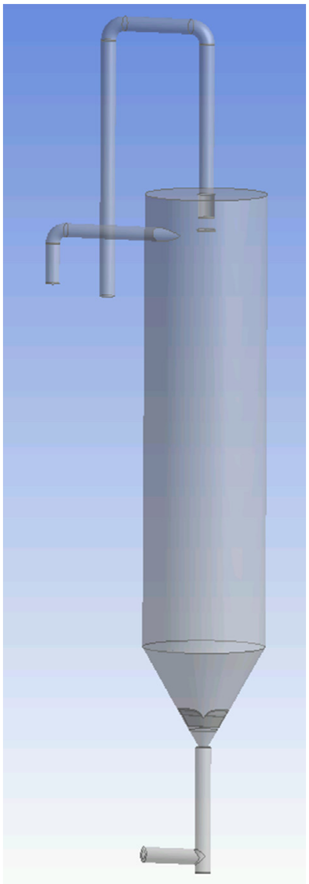

Fig. 1. Schematic of modelled domain. Both the top and tangential feed lines can be seen. However, only one feed line is active at any time. Note that when the top feed inlet is used there is a deflector plate suspended below the feed nozzle which stops flow from short-circuiting directly to the outlet.

purposes and then compare simulated flow-fields with the experimental data, first for the top-fed vessel and then for the vessel with a tangential inlet. Then the residence time distributions are presented together so that comparison based on the different methods, geometries and turbulence models can be made. The remainder of the paper is presented as follows: Section 2 describes the experimental data used for model validation; Section 3 describes the CFD model used in this study; Section 4 presents the simulation results and comparison with the experimental data and Section 5 contains the conclusions.

2. Experimental study

Alcoa commissioned CSIRO Mineral Resources Flagship to generate the experimental data used in this study. A laboratory-scale ($\sim$1.5 m tall and $\sim$0.4 m diameter) model of a full-scale digester was used in the experiments. Water at ambient temperature was used as the working fluid. The data reported here are for an inlet flow rate of 127 L min^{-1}.

Superficial residence time is a crude estimate of how long a fluid particle will be present within a vessel, and is defined as the volume of the domain divided by the volumetric flow rate through it. In the cases considered here, the superficial residence time, T_o, is $\sim$88 s. The results from the experiments and simulations will be compared using a normalised time (measured time, t, divided by T_o).

Laser Doppler Velocimetry (LDV) was used to measure the velocity components along a line from the central axis to just off the vessel walls. These readings were taken at several vertical elevations – those taken at 415 mm and 1465 mm above the cone are reported here as they are representative of the behaviour observed at all locations. The measurement device was held stationary at each point along the radius for approximately two minutes, with 2000 readings taken in each period.

Residence Time Distributions (RTD) were determined using a salt tracer. A pulse of salt solution was injected into the inlet pipe once the flow had settled down and the concentration of salt was measured at the outlet over time.

3. Model description

3.1. Turbulence modelling

Different turbulence modelling approaches were tested in an effort to determine the most accurate method for calculating the flow field using the most efficient computational approach. If steady simulations can be performed using a RANS turbulence model the computational requirements, in terms of mesh size and computational run time, are vastly smaller than if a transient scale-resolving approach is needed. This knowledge clearly has a huge impact on the computational resources required to perform such analyses. We note here that although a RANS model may give a steady solution this does not mean the real flow is steady. Rather it says that the unsteadiness can be removed by the Reynolds-averaging process and represented by a simple eddy diffusion hypothesis and there is no underlying transient flow. As flow separation is not a governing feature of the flows, wall functions are considered to be sufficient to predict flow in the wall region.

Previous work (Brown and Fletcher, 2005) had shown that the standard k-ε model (Launder and Spalding, 1974) could predict the flow in the top-fed digester but was unable to predict the behaviour in the tangentially-fed vessel. The k-ε model yields a steady-state solution, where the experiments showed large-scale unsteadiness of the flow. The two-equation SST (Menter, 1994) and Reynolds Stress Model (SSG RSM) Speziale et al., 1991 were also tested in this work. Finally, given the success in predicting unsteady flow in laboratory-scale precipitators (Brown et al., 2014), the Scale Adaptive Simulation (SAS-SST) model of Menter and Egorov, 2010 was considered. When applying the SAS-SST model if the mesh is sufficiently fine the large-scale turbulence structures can be captured if the flow is globally unsteady (Menter and Egorov, 2010). If the flow is not globally unstable, i.e. there is nothing to trigger the generation of large-scale turbulence structures, the simulation model may remain in RANS mode and no structure is resolved. In order to avoid this, a zonal LES region was introduced in the inlet feed pipe in order to convert modelled turbulence into resolved turbulence which ensures that the SAS-SST model will behave in scale-resolving mode and continue to resolve the large-scale turbulence structures in the downstream region. This initiation method uses a harmonic turbulence generator to convert modelled turbulence kinetic energy into random fluctuations which trigger the scale-resolving mode (Menter et al., 2009). In addition the curvature correction term was enabled in the SAS-SST simulations to avoid artificial decay of vortical structures (Smirnov and Menter, 2008).

3.2. Computational domain and mesh

The meshes used in this study were informed by the experience in modelling alumina precipitators (Brown et al., 2014). The tangentially-fed digester is inherently 3-dimensional, so no attempt was made to reduce the geometrical complexity.

Please cite this article in press as: Whyte, D.S., et al. Predicting flow and residence time in alumina digestion vessels. Chem. Eng. Sci. (2016), http://dx.doi.org/10.1016/j.ces.2016.12.036