NUMERICAL MODELLING AND VALIDATION OF SHEAR THINNING FLOWS IN A LAB-SCALE DIGESTER

S. Nagasundaram (s.nagasundaram@utwente.nl), A.K. Pozarlik (a.k.pozarlik@utwente.nl), H. Norouzi-Firouz (h.norouzifirouz@utwente.nl), G. Brem (g.brem@utwente.nl)

Thermal Engineering, University of Twente, P.O. Box 2017, 7500 AE, Enschede, The Netherlands

Abstract

Anaerobic digestion is a process of biochemical conversion of bio-waste into a fuel-gas mixture (methane+carbon dioxide) by bacteria, in the absence of oxygen. The process is carried out in a tank which is referred to as a digester and the efficiency of the biochemical waste conversion process is dependent on several factors, with mixing being one of the key factors.

A typical digester measures up to 20 meters in height and diameter and if the role of mixing in the intensification of biogas production in a full scale digester is to be studied, the numerical models first need to be verified. In this paper, the available CFD numerical models are first applied to model a shear-thinning flow in a lab scale digester and the flow variable data from computations are compared against available PIV measurements for the same digester. The flow variables considered in the validation are the turbulent kinetic energy and the mean velocities. The RNG k-epsilon model is found out to be the most suited turbulence model to simulate shear-thinning and the Multiple Reference Frame (MRF) approach was found to be fairly accurate and computationally economical in predicting flow trends.

KEYWORDS: Anaerobic Digestion, CFD, Numerical Modelling, Shear thinning flows
1.) **Introduction**

Anaerobic digestion is an interesting option to process organic bio-waste and generate green biogas, with minimal energy expense, since there no oxygen pumping required and because the process happens at low temperatures. The process of anaerobic digestion thus serves as an economical and an easily-employable technological solution in the process of transitioning towards biofuels and green energy. Nevertheless, the maximum operation temperature of the process is between 50-60 °C and being a process carried out in low-temperature, it involves big timelines, typically lasting weeks per digestion batch. Thus, even though anaerobic digestion is an economically and technologically viable option, the high digestion times remain a constraint.

The anaerobic digesters have a mixing system to constantly agitate the bio-waste/water mixture (sludge) to distribute the nutrients and bacteria uniformly, in the digester and to avoid solid sedimentation. There are several industrial techniques used to agitate the mixture, with mechanical mixing and gas/sludge recirculation being the most commonly used technologies. The mechanical mixing has been reported to be the most efficient in terms of energy consumed per unit volume [4].

In case of a mixing system with an impeller, the presence of the manure particles in the sludge being digested yields a shear thinning property to the fluid. This leads to the production of caverns in the digester, in the regions close to the impeller. This in turn makes the mixing patterns non-uniform and leads to the presence of low-velocity zones in the digester, which makes the process non-uniform.

The process performance is dependent on an array of factors feed characteristics, feeding rate, local pH of the mixture, temperature, bacteria content and mixing characteristics. The role of mixing in the production of biogas is often confusing and conflicting trends have been reported in literature. For example, some papers report continuous mixing to be a waste of energy and that intermittent mixing is more efficient [9,12]. Some papers claim that continuous mixing is vital to distribute the nutrients and bacteria uniformly and that it improves biogas production, especially in digesters with high total solid percentages [7]. High strain rates due to mechanical mixing have been proven to be detrimental to the process.

The CFD modelling of the anaerobic digestion process could typically be divided into two parts, namely the hydrodynamic modelling and the biochemical modelling. In order to be able to predict biogas outputs, both these parts need to be integrated and validated against experiments. The opacity and the texture of the sludge already make it a challenging task to measure flow variables for validating CFD and even more so, in a full scale digester. Thus in order to understand the flow dynamics, a synthetic transparent sludge, whose characteristics are similar to the real sludge, was agitated in a lab scale digester and visualized by carrying out PIV measurements, by Sindall et. al. [10].

In this work, effort was taken to model the hydrodynamic behavior of a continuously stirred, lab-scale anaerobic digester and to validate the different turbulence and rotation models available in CFD. Flow variables trends predicted by CFD were compared to the PIV measurements reported for the same digester as reported in Sindall et. al. [10].

2.) **Research Methodology**

2.1) **CFD and turbulence modelling**

The simulation package used in this work is Ansys Fluent 18.2. The flow is solved using the Finite Volume Method. The applied equations are the continuity equation, as described in Equation (1), the momentum equation, as described in Equation (2) and the transport equations for turbulent quantities turbulent kinetic energy, $k$ and turbulent dissipation $\varepsilon$, are described in Equation (3) and (4) respectively,
for the RNG k-epsilon turbulence model. In Equations (3) and (4), $G_k$ represents the generation of turbulent kinetic energy due to mean velocity gradients. $G_B$ represents the generation of turbulent kinetic energy due to buoyancy. $Y_M$ represents the contribution of the fluctuating dilatation in compressible turbulence due to overall dissipation rate. The quantities $\alpha_k$ and $\alpha_\varepsilon$ are the inverse Prandtl numbers for $k$ and $\varepsilon$, respectively. $S_k$ and $S_\varepsilon$ are the user-defined source terms. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are model-specific constants.

The energy equation is not solved, since the temperature is assumed to be maintained constant at 35°C.

\[
\frac{\partial (\rho)}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]  

(1)

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = - \frac{\partial p}{\partial x_i} + \frac{\partial (\tau_{ij})}{\partial x_j} + \rho g_i
\]  

(2)

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j} \right) + G_k + G_B - \rho \varepsilon - Y_M + S_k
\]  

(3)

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} \left( G_k + C_{3\varepsilon} G_B \right) - C_{2\varepsilon} \rho \left( \frac{\varepsilon^2}{k} \right) + S_\varepsilon
\]  

(4)

The three directions of the coordinate space, namely x, y and z are represented by $x_i$. The components of the velocity in these three directions respectively are represented by $u_i$, where the index $i$ takes the values of 1, 2 and 3 respectively. The stress tensor for the fluid is described by the term $\tau_{ij}$ in the tensor notation and the $g_i$ represents the momentum source term in the same $i$ direction.

Since, the presence of a rotating impeller makes the flow regime turbulent, RANS turbulence modelling is employed. Four turbulence models were tested namely, RNG k-epsilon, Realizable k-epsilon, Reynolds Stress Model and k-omega SST. The transport equations for the RNG k-epsilon are documented above and for the other models, they are documented in the Fluent User Guide manual [1].

The sludge being mixed consists of three phases in a real-case scenario i.e. a.) Solid manure particles, b.) Liquid water in the sludge, c.) Fuel-gas produced by anaerobic digestion. Thus modelling the solid, liquid and gas phases together requires a multi-phase Euler-Euler approach. It is both computationally expensive and requires a thorough closure model treatment in order to correlate the momentum and turbulence exchange between the phases in the Navier-Stokes equation.

The manure particles in the sludge being digested, yield a shear thinning property to the fluid and thus solid-liquid mixture can be described as a single fluid, exhibiting Non-Newtonian behavior. This approach has been used previously in literature [13,14]. This reduces the need to model the solid and liquid phases separately and reduces the computational time. The effective viscosity of the sludge, could be modelled using the Non-Newtonian Power Law equation below.

\[
\mu_{\text{eff}} = K \left( \frac{\partial u}{\partial y} \right)^{n-1}
\]  

(5)

where $K$ is the Consistency index and $n$ is Power law index.

The liquid being modelled is a 0.5 g/L CMC (carboxy methyl cellulose) solution, whose rheology was calculated in [10]. The power law parameters as modelled in the work is tabulated below.
### Power Law Parameters

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Consistency index $K$</td>
<td>$0.0707 \ Pa s^{0.6052}$</td>
</tr>
<tr>
<td>Power law index $n$</td>
<td>0.6052</td>
</tr>
<tr>
<td>Allowable viscosity range</td>
<td>0.004-0.012 kg/m-s</td>
</tr>
</tbody>
</table>

*Table 1. Power Law Parameters [10]*

#### 2.2) Geometry

A cylindrical vessel that has a diameter of 200 mm and a height of 200 mm is modelled. The tank has four baffles equally spaced around the tank and has a four blade impeller. The impeller diameter is 90 mm and it has a height of 20 mm. All the blades are flat and have a thickness of 8 mm. The baffles have a thickness of 10 mm and extend 10 mm inwards from the tank wall. The computational domain and the impeller are shown in Fig. 1 and Fig. 2, respectively.

![Computational domain and impeller geometry](image)

*Figure 1. Computational domain and impeller geometry*

#### 2.3) Boundary conditions and solver settings

In order to capture the rotation of the impeller, two different models have been commonly used in literature, namely the Multiple Reference Frame (MRF) model and the Sliding Mesh model [3, 11, 13, 14]. These methods are discussed in detail in the Fluent user guide [1]. Like depicted in Fig. 1, the region around the impeller represented as MRF, was designated as the moving region and rotational speed was assigned. The impeller rotation speed is 100 rpm.

The top of the tank was assigned as a free surface and the rest of the surfaces were designated as no-slip walls.
Residual criteria of $1 \epsilon^{-4}$ was set and the default relaxation factors were used. Second order discretization was used for flow variables and turbulence quantities. First order scheme was used for temporal discretization in case of transient simulations.

2.4) Meshing

To capture the effects of impeller rotation, the MRF region was meshed finer than the tank region. The finest mesh had 2 million elements and the coarsest 280,000 elements. In between, two other meshes were studied, as well. Furthermore, the transient effects were assessed by comparing the transient and the steady state calculations. In order to avoid solution dependent results, a mesh independency study was performed. The results showed that a mesh with 1 million elements was sufficient for computations.

3.) Results and discussion:

In order to compare the results of CFD to measurements a circular surface extending from the bottom to the top of the tank is chosen at $r/R= 0.6$. The line of measurement is presented in Fig. 2 (marked as LOM). The line is located centrally between two baffles in the circumferential direction.

The velocity and kinetic energy profiles are plotted along the line of measurement from bottom to top of the tank. The comparison of the results obtained from both approaches against PIV measurements is shown from Fig. 3 till Fig. 8. On the y axis, the non-dimensional height $y/H$ is taken. The flow variables that were monitored are the mean velocities $u$ and $v$, in x and y direction respectively and the turbulent kinetic energy $k$.

![Figure 2. Measurement locations](image)

3.1) Effect of applying different turbulence models for MRF steady state approach

In order to be able to capture the hydrodynamics in the digester accurately, especially in a shear-thinning flow, it is key to capture the turbulence accurately, so that the velocity gradients are well-solved. Thus a study was carried out to check the applicability of the turbulence models. The MRF steady state approach was used in all the cases. The mean velocity and turbulence trends as predicted by different turbulence models are shown below along with measurements from PIV, for comparison.
The k-omega SST model was assessed to be unreliable in predicting both the turbulence and the velocity trends, as represented above, from Fig.3 till Fig. 5. The RSM model was the best in resolving the turbulence peak, even though the velocity trends are not predicted accurately. Additionally, it requires an excessive computational time by adding 4 more equations to solve, in comparison to the two-equation turbulence models.

The RNG k-epsilon model was thus selected to be the most suited model for predicting the velocity and turbulence fields, since it has a low computational time and can overall predict the trends with good accuracy.
3.2) Investigation of steady/transient simulation approaches

As mentioned above, the steady and the transient simulation approaches were investigated, in order to be able to study particle residence times and the sedimentation process.

In order to keep the results consistent, the study was done using a single standard mesh and turbulence model for all the cases, the 1 million elements mesh and RNG k-epsilon model were chosen for this study, based on the previous results.

The Sliding Mesh approach is transient in nature and therefore the mesh was physically rotated every timestep and has to be iterated progressively, in order to account for the rotation of the impeller. In the MRF approach, there is an option to choose both steady and transient approaches, since the velocities are added to fluid by including a Coriolis component in the momentum equation.

In case of transient simulations, a total physical time of 30 s and a timestep of 0.02 s was selected.

The trends for the mean velocities and the turbulent kinetic energy, as predicted by the investigated approaches studied, namely a.) MRF steady, b.) MRF transient and c.) Sliding mesh transient, are represented below from Fig.6-8.

![CFD/PIV comparison](image)

*Figure 6. Mean velocity $u$ for different rotation approaches and RNG k-epsilon*
As seen from the above figures, the Sliding Mesh model is seen to provide slightly more accurate trend-line prediction of the velocity field in comparison to the MRF model. However it underpredicts the turbulent kinetic energy peak.

The steady state approach gives good predictions regarding the velocity and turbulence trends. The steady and transient MRF approaches present similar results and thus prove that the MRF steady state approach can serve as a good first approach in order to predict the velocity and turbulence trends with good accuracy and acceptable computational time. In order to compare the times taken by the simulations, the results are summarized in the table below.
<table>
<thead>
<tr>
<th>Case description</th>
<th>Method of rotation</th>
<th>Physical time simulated/ Timestep</th>
<th>Computation time</th>
</tr>
</thead>
<tbody>
<tr>
<td>MRF steady</td>
<td>Multiple Rotating Frame</td>
<td>-</td>
<td>~4 hours</td>
</tr>
<tr>
<td>MRF transient</td>
<td>Multiple Rotating Frame</td>
<td>30 seconds/ 0.02 s</td>
<td>~6 hours</td>
</tr>
<tr>
<td>Sliding mesh</td>
<td>Sliding Mesh</td>
<td>30 seconds/0.02 s</td>
<td>~168 hours</td>
</tr>
</tbody>
</table>

*Table 2. Computational times taken by different temporal approaches*

4.) **Conclusions**

A numerical model to study the mixing process in anaerobic laboratory scale digester was developed and validated. In terms of turbulence, the RNG k-Epsilon model was found to be the most reliable in terms of predicting both the velocity and turbulence trends simultaneously. The k-omega SST model provided unreliable results in terms of the velocities and RSM model was found to be computationally expensive and less accurate than the RNG k-epsilon model.

The results show that the Sliding Mesh model in general gives better estimation of the turbulence field compared to the MRF model. However, the MRF approach is computationally less time consuming and thus less expensive. Therefore for any parametric research the MRF model is advised, whereas the sliding mesh approach is superior for in-depth study or the final assessment of the parametric research.

5.) **Acknowledgements**

This work was carried out at the Thermal and Fluid Engineering Group, University of Twente, as part of RVO-funded project “Delen Maakt Meer”. Authors would like to thank Host Bioengineering B.V. and Saxion Institute of Applied Sciences, Enschede, for their support and all the fruitful discussions during the project meetings.

**References**

1.) Ansys Fluent User guide,  


