# Mixing analysis of a non-Newtonian fluid stirred with a hydrofoil impeller: Particle Image Velocimetry vs Computational Fluid Dynamics

**David Fernandes del Pozo**[1], **Alain Liné**[2], **Kevin M. Van Geem**[3], **Ingmar Nopens**[1]

[1] BIOMATH, Department of Data Analysis and Mathematical Modelling, Ghent University, Coupure Links 653, B-9000, Ghent, Belgium, david.fernandesdelpozo@ugent.be, ingmar.nopens@ugent.be
[2] LISBP, Université de Toulouse, INSA, INRA, CNRS, Toulouse, France, alain.line@insa-toulouse.fr
[3] Laboratory for Chemical Technology, Technologiepark 914, B-9052, Ghent University, Ghent, Belgium, kevin.vangeem@ugent.be

**Abstract:** A detailed hydrodynamic study using Particle Image Velocimetry (PIV) is carried out to understand the flow behavior of viscous fluids at different flow regimes. A non-Newtonian fluid (Carbopol) mimicking the rheological behavior of anaerobically digested sludge is stirred with an A310 hydrofoil impeller in a 70L baffled vessel. Results show a complex 3D hydrodynamic structure under different flow regimes and at different spatial locations. The PIV data-set is used to evaluate the accuracy of the calibrated rheological model and the performance of different modelling approaches using laminar and different turbulence models in terms of mean and fluctuant quantities using OpenFOAM.

**Keywords:** Carbopol, mechanical mixing, anaerobic digestion, OpenFOAM, calibration, model assessment

## Introduction

Anaerobic Digestion (AD) has become an attractive process technology in Water and Resource Recovery Facilities (WRRF) in the last decade because it enables to valorize sewage sludges into methane-rich biogas. Since renewable energy resources technologies are in integral part of several EU environmental policies, AD reactors are being currently implemented to treat a broader selection of substrates ranging from agricultural crops, manures, and vegetable by-products. In order to ensure the correct digestability of the incoming sludge, mixing is of primary importance to prevent stratification and acidification that might be linked with a reduction in biogas production or process shutdown. Most of the WRRFs in Europe employ Continuously Stirred Tank Reactors (CSTR) with hydrofoil impellers. Since digesters are usually over-designed to account for their non-Newtonian behavior, its mechanical energy consumption constitutes one of the most energy intensive operations in WRRFs. Therefore, a deeper understanding is required to analyze the flow of such viscous slurries inside the digester to prevent stratification and to minimize dead volumes.

This work studies the hydrodynamics encountered in viscous fluids in detail using a flow visualization technique (PIV). Finally, the hydrodynamic data is compared to a Computational Fluid Dynamics (CFD) model to evaluate the rheological model and different turbulence models for different flow scenarios.

## Materials and Methods

### Mixing tank

The tank used in this study consists of a standard cylindrical vessel equipped with four equally spaced baffles (width B = 0.045 m = T/10). The vessel is made of poly(methyl methacrylate) and had a diameter and a liquid height of T = H = 0.45m. The cylindrical tank is placed in a cubic tank filled with water to minimize optical refraction. The impeller used in this study is a standard A310 hydrofoil impeller with D = 0.15m.

### Fluid

The tank is filled with ≈70L of Carbopol 980 (0.06%w/w, Sigma Aldrich) mixed with high purity water. Carbopol is a transparent fluid model selected because it is rheologically similar to digested sludge [1] and displays shear thinning behavior as well as an apparent yield stress. The concentration of Carbopol is adjusted to behave as similar as possible to a 4% TSS of digested sludge. The Carbopol flow curve is obtained from a Thermo Haake rheometer with a Cone and Plate (2º) geometry. A local optimizer is used for calibration of the Herschel-Bukley model and the Generalised

Likelihood Uncertainty Estimation (GLUE) method is used to derive the confidence intervals on the calibrated parameters.

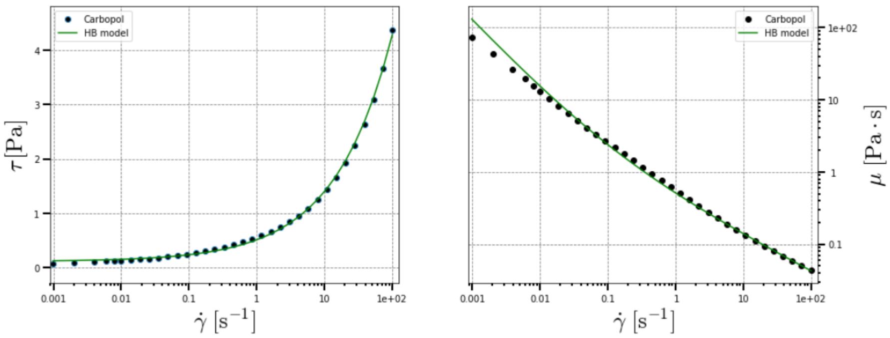


*Figure 1: Flow curve of Carbopol at 20ºC with a calibrated Herschel-Bulkley (HB) model.*

### PIV measurements

The experiments were carried out with a Nd:YAG laser (Dantec Dynamics, λ=532nm), a FlowSense EO 4M camera (native 7.4μm/pixel), and Rhodamine-coated seeding particles (1-20μm, thus guaranteeing a sufficiently small relaxation time). Raw data processing (1000 & 2500 snapshots) was made with Dantec Dynamic Studio 2015a and MATLAB code for all post-processing of PIV data.

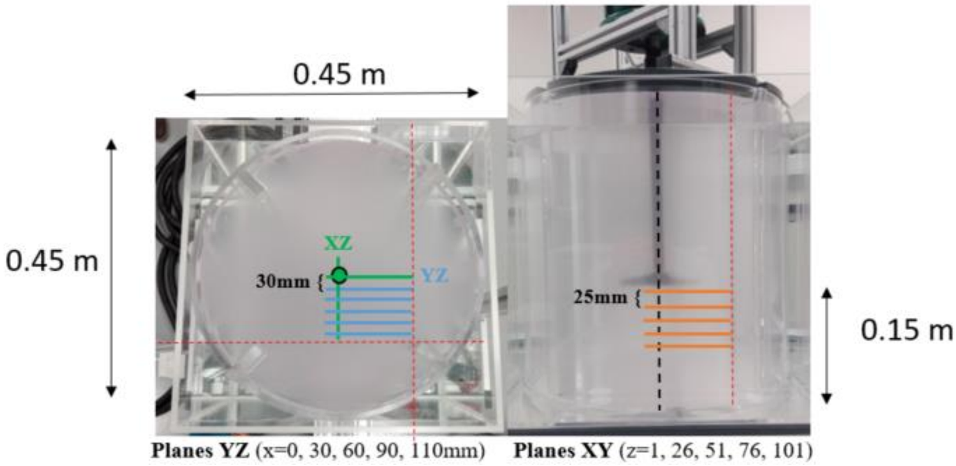


*Figure 2: Dimensions and positions of the laser sheets with respect to the impeller (XZ, YZ, XY). The red dotted lines indicate the maximum depth of field imposed by the baffles.*

*Table 1: Overview of PIV experiments*

| Plane | Coordinates from Impellers position (mm) | RPM (clock-wise) |
|---|---|---|
| XZ | 0 (reference plane) | 50, 100,250, 500 |
| YZ | 30, 60, 90, 110 | 100, 250, 500 |
| XY | 1, 26, 51, 76, 101 | 100, 250, 500 |

### CFD model

OpenFOAM v5.0 was used to solve mass and momentum equations for laminar and turbulent flow of Newtonian and non-Newtonian fluids using *simpleFoam* (steady state solver) with the Multiple Reference Frame (MRF) approach, and with 2$^{nd}$ order schemes for all terms. Additionally, different 2-equation models for turbulence modelling are tested in turbulent flow scenarios. Simulations were considered converged when normalised residuals were below $10^{-4}$ for all variables and the impeller torque achieved a constant value. Post-processing of CFD data was done with ParaView v5.4.0 and Python Jupyter Notebook.

## Results and Discussion

### Mean velocities

Figure 2 shows the different hydrodynamic structures in a Horizontal-Vertical plane (XZ) for different 50 and 250 RPMs as obtained by the PIV. Results indicate the strong influence of viscosity on the development of the characteristic downward jet induced by the hydrofoil impeller.

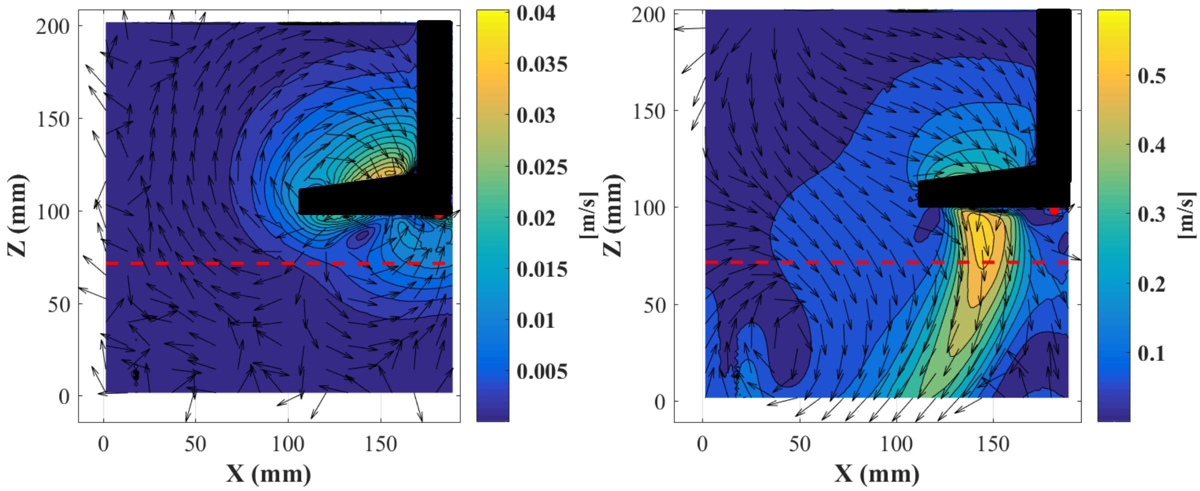


*Figure 3: Mean velocity contour and normalized velocity vector plots for plane XZ (y=0mm) at 50 [left] and 250 rpm [right]. The red dot indicates the location of the center of the impeller's shaft and the red dashed line indicates an intersection with a XY plane 26mm below the impeller. Note that direct comparison between contour colors is not possible.*

## Intersection of PIV planes

Radial shared mean ($\bar{u}$) and averaged squared fluctuating velocity $\left(\overline{u_{fluc}{}^2}\right)$ components are given in Figure 4 at the intersection of PIV planes with the reference plane XZ (y=0mm) for 250 RPM. The spatial accuracy of the PIV planes is very good since shared velocity quantities for two intersecting planes are almost identical.

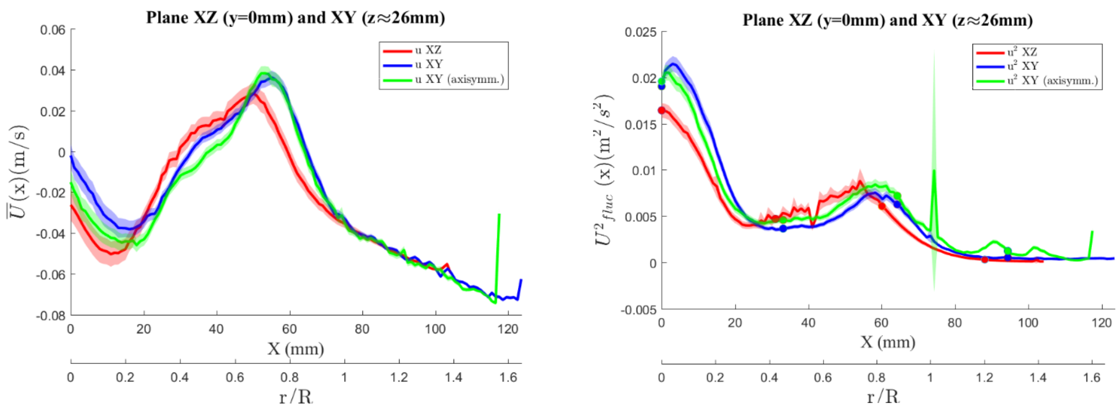


*Figure 4: $\bar{u}(x)$ [left] and $\overline{u'^2}$ [right] at the intersection between plane XZ (y=0mm) and XY (z≈26mm). Lower x-axis is normalised with the impeller's radius (75mm). 95% CI (Confidence Intervals) are plotted for all variables.*

## PIV vs CFD

In order to verify that OpenFOAM is suitable to conduct the modelling analysis, the CFD model performance is evaluated against PIV data from literature concerning the same geometry with water as a fluid (Bugay, 2002). Figure 5 shows that the $k$-$\varepsilon$ turbulence model is able to capture the main velocity trends for the three components with reasonable accuracy. Additionally, the computed torque (the sum of viscous and pressure moments exerted over all impeller surface cells) was used to compare the average dissipation rate of turbulent kinetic energy compared to the experimental value ($\varepsilon_{CFD} =$ 0.0128 W/kg, $\varepsilon_{exp} = 0.0114$ W/kg). Next, the CFD model is compared against the new PIV data for the laminar (Figure 6) and turbulent scenarios (results not shown in the abstract) using Carbopol as fluid.

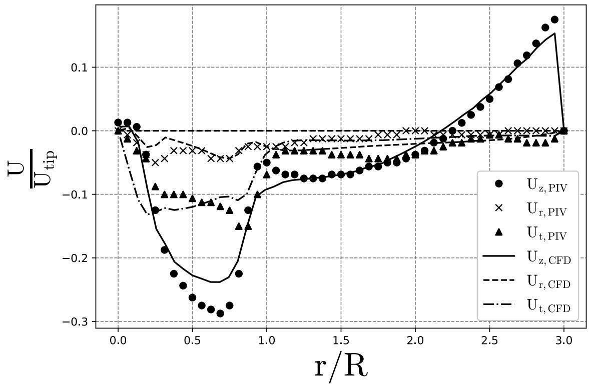


*Figure 5: Comparison of experimental PIV data at 200RPM from Bugay (2002) and CFD results for dimensionless axial (z), radial (r), and tangential (t) velocity components along the dimensionless tank radius* $(R = 0.075m)$ *at 5 mm below the impeller with water. The k-ε turbulence model was used in the CFD simulations.*

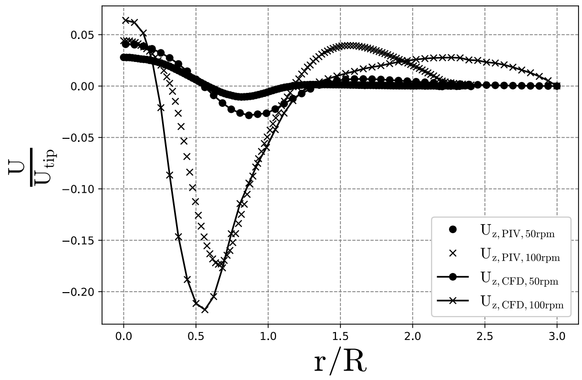


*Figure 6: Comparison of experimental PIV data at 50 and 100RPM and CFD results for dimensionless axial (z) velocity along the dimensionless tank radius* $(R = 0.075m)$ *at 26 mm below the impeller with Carbopol. The CFD simulations were run in laminar flow with calibrated rheological parameters from Figure 1.*

As can be seen in Figure 6, the simulation of non-Newtonian fluids is more challenging even at laminar flow regimes. It is evident that the flow fields are highly sensitive to the stirring velocity due to the difference in local viscosities arising from a heterogeneous shear rate field.

## Conclusions

A detailed hydrodynamic analysis is performed for the mechanical mixing of a viscous fluid, surrogate for anaerobic digestion. Analysis of the flow induced by the impeller revealed a complex 3D hydrodynamic structure that is highly dependent on the impeller's revolution. A database in laminar, transitional, and turbulent flow of a non-Newtonian fluid is obtained and is available for further studies, and for CFD benchmarking of models. In turbulent flow, the three components of the mean flow and the nine components the Reynolds stress tensor are available at different points below the impeller.

As future perspectives, the database will enable the possibility to 1) analyse the presence of caverns in laminar flow, 2) local analysis of the trailing vortices induced by the hydrofoil impeller's velocity fluctuations, and 3) analysis of local shear rate and dissipation rate of turbulent kinetic energy for a non-Newtonian fluid.

## References


[1] Eshtiaghi, N., Yap, S. D., Markis, F., Baudez, J. C., & Slatter, P. (2012), *Water Research*, *46*(9), 3014–3022

[2] Bugay, S., Escudié, R. and Liné, A. (2002), Experimental analysis of hydrodynamics in axially agitated tank. *AIChE Journal*, 48: 463-475.