

Ultrasound Imaging Velocimetry measurements and Large Eddy Simulation of domestic slurry in pipe bends

G. Oldenziel^{1,2*}, A. Dash^{4,5}, M. van der Eijk¹, T. Boschetti¹, A. Talmon⁷, A. Zubova⁶, F. Clemens³

1: Deltares, The Netherlands

2: Laboratory for Aero & Hydrodynamics

Delft University of Technology, The Netherlands

3: Skills in Motion BV, The Netherlands

4: Multiphase Systems, Process & Energy, Department of Mechanical Engineering, Delft University of Technology

5: KWR Water Research Institute, Nieuwegein, The Netherlands

6: Pascal Technologies, Norway

7: Section offshore and dredging engineering, Delft University of Technology, The Netherlands

* Correspondent author: gosse.oldenziel@deltares.nl

Keywords: UIV, LES, domestic slurry, non-newtonian fluids, pipe flow, hydraulics, pipe bend

INTRODUCTION

In this paper we present ultrasound imaging velocimetry (UIV) measurements and Large Eddy Simulation (LES) of the flow of a water clay mixture in a prototypical pipeline system component: a 90 degree pipe bend. The background of the research are so-called future sanitation systems, where water usage is more efficient than in conventional sanitation systems due to the increased concentration of the domestic slurry and local (in the neighborhood) recovery of useful constituents from the domestic slurry is promoted. For the design of these networks, knowledge of the flow of these, typically non-Newtonian, fluids with a finite yield stress is needed, see Thota Radakrishnan et al. (2018). As a starting point, we compare the mentioned flow observations and simulations of a water-clay mixture in a pipe bend. We are interested in the quality of the characterization of this complex, non-optically accessible flow, that can be obtained via UIV measurements and LES simulations, their agreement, the performance of the present implementation of LES for the relevant non-Newtonian fluid and other possible modelling effects in the LES. The last point of interest that will be touched upon is the recovery of the velocity profile downstream of the bend. Experimental and numerical analysis of the pressure losses in the bend will be performed at a later stage.

EXPERIMENTAL SETUP AND NUMERICAL SETUP

Experimental setup

The experimental setup is shown in Figure 1. The setup consists of a reservoir where a water clay mixture is mixed and from where it is pumped into a long ($> 800 D$) straight pipe with an inner radius of 0.0508 m using a centrifugal slurry pump (NT 3153, Xylem) via pressurized flow. Upstream in this straight section is the electromagnetic flow meter (Optiflux 2300c, Krohne). The flow is directed upwards via a 90 degree bend (Van de Lande b.v., $R/D = 1.5$), downstream of which is a vertical pipe segment of $22 D$ and a second identical 90 degree bend directs the working fluid back towards the reservoir, where it enters the mixing tank, via a partially downwards sloping section. Five pressure sensors (PDCR5031, GE) are installed in the pipe and two temperature sensors are installed as indicated in Figure 1. There is one bypass section immediately downstream of the pump. At several locations valves are installed and the pipe material is polyvinyl Chloride (PVC) with a wall thickness of 4.2 mm.

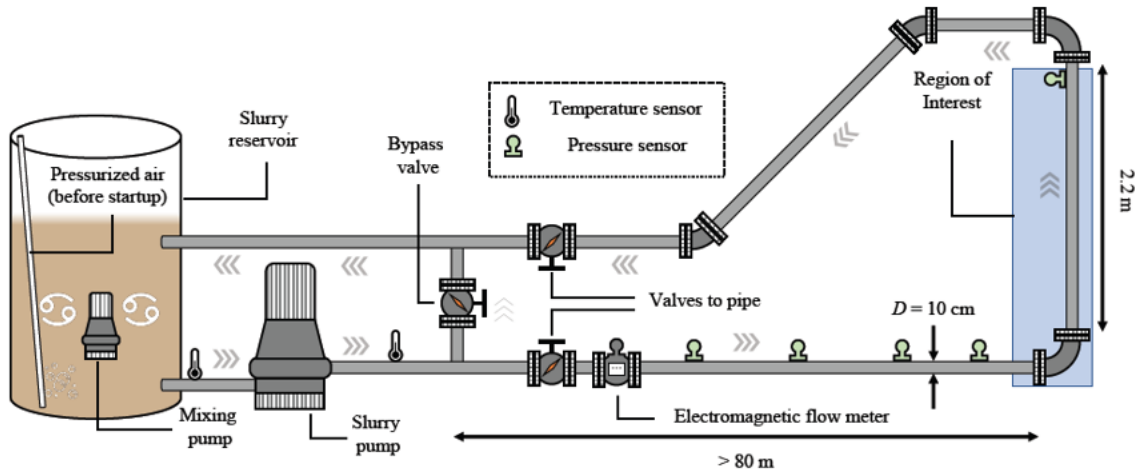


Figure 1: Beta Loop slurry flow facility at Deltares, The Netherlands. Modified from Dash (2022).

The working fluid is a water kaoline clay mixture (21% w/w or 9% v/v) that is described by a Herschel Bulkley model, given by $\tau = \tau_y + K \dot{\gamma}^n$, where τ_y is the yield stress, K is referred to as the consistency index, and n is the behaviour index. The mixture is heavier than water and shear thinning ($n < 1$), see Table 1, where the density is denoted by ρ . For the description of the quantification of the rheology the reader is referred to the Supplemental material in Dash & Poelma (2022). This mixture is used because it can be seen as a model for domestic slurry, see Thota Radakrishnan et al. (2018). The flow rate for which the flow through the 90 degree bend will be analysed corresponds to $U_b = 1.48$ m/s. This flow rate is selected because no settling of clay particles occurs and the system thus steadily operates over long times (Dash & Poelma 2022). This flow rate is attained by closing the bypass valve completely and operating the pump at 20 Hz. A comparison of the rheogram as measured in a rheometer (HAAKE MARS I, Thermo Scientific, cup bob geometry) is shown in Figure 2. The pseudo shear rate for the current pipe flow ($= 8U_b/D$) is 118 s^{-1} , so this is in the range where the HB model and the measurement data are virtually indistinguishable.

Table 1: fluid parameters from rheometer quatification and flow parameters.

Q	696	l/min
U_b	1.48	m/s
ρ	1152	kg/m^3
τ_y	0.8889	Pa
K	0.1579	$Pa \text{ s}^{0.1579}$
n	0.4579	—

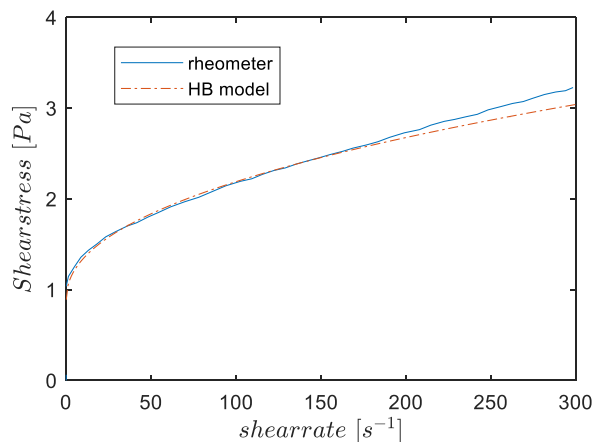


Figure 2: Shear stress versus shear rate of water-clay sample taken under turbulent conditions.

Ultrasound Imaging Velocimetry (UIV) (see Poelma 2017) is a technique in which images, as constructed via analysis of ultrasound transducer arrays, are analysed using particle image velocimetry (PIV) (see Adrian & Westerweel 2010) algorithms. It is thus a quantitative imaging technique that can be used even when the fluids (and container solids like PVC) are not optically transparent. Data for UIV were acquired with a SonixTOUCH Research (Ultrasonix/ BK Ultrasound) device. The UIV data that is presented in this work is taken from measurements in three measurement planes. All three planes image a radial-axially oriented plane, referred to as ‘top’ for the vertically oriented plane from the top of the pipe to the center, ‘center’, for the horizontal plane from the centre of the pipe to the side, and ‘bottom’ for the vertically oriented plane from the center of the pipe to the bottom (note that due to structural obstructions this plane is not exactly vertical), see Figure 3. The sensor head (linear array transducer (L14-5/38) of 128 elements with a pitch of 0.3 mm and an elevation focus at 16 mm) emits ultrasound pulses with centre frequency at 5 MHz (two cycles) and samples the reflected signal at 40 MHz. The beam sweep direction is set opposite to the flow direction to reduce the particle displacement (Zhou et al. 2013) The focus is set at 0.045 m and the imaging depth at 0.06 m and the sector setting is set to 50%. When activating 64 transducer elements the attained frame rate is 287 frames per second. The speed of sound in the domestic slurry was measured to be 1480 m/s in a separate slurry sample. The mean is subtracted from the ultrasound signal and the result converted to images (commonly referred to as B-mode images) using the speed of sound, where the spatial resolution in radial direction (from pipe perspective) is down sampled by a factor 4. The typical image size is 203×33 pixels (axial $\times$ lateral). We perform single pass cross-correlation processing using windows of 16×24 pixels (axial $\times$ lateral) with 50% overlap in axial direction and 85% overlap in lateral direction resulting in three columns of displacement vectors of which one column that is not at the edge. In total five discontinuous sets of 2503 images are collected in this way, so in total 43.6 s of data).



Figure 3: left: orientations of measured velocity profiles, right: detail of experimental setup.

The vector results are processed differently for average velocity profile results as for the quantification of the turbulence. To obtain average velocity profiles correlation averaging is applied (Adrian & Westerweel 2010) over all 12515 images of all sets, and a 5×5 pixel 2D Gaussian peak fit is used. To obtain time varying velocities we apply sliding correlation averaging: we average over ten correlation planes and take a step of five, so for example we take separate average images 1-10 and cross correlate these with the average of images 6-15, then we correlate the average of image 11-20 with the average image of images 16-25 etc. We determine the correlation peak location at sub-pixel accuracy via a traditional 3-point 1D Gaussian peak fit. We apply backward sweep correction (Zhou et al. 2013). To quantify the turbulence for the flow in the upstream straight pipe section, we apply the space-time filtering as described in Appendix A in Dash et al. (2022). The turbulence intensity (TI) is approximated via the expression:

$$TI = \frac{\sqrt{\frac{1}{3} \cdot (u'^2 + 2v'^2)}}{U_b} \cdot 100$$

where primes denote the instantaneous velocity fluctuations obtained by subtracting the mean velocity from the velocity signal. UIV measurements are taken at multiple locations, namely: -17 D (far upstream), +1 D, +4 D, +7 D, +10 D, +13 D, +17.5 D and, +21 D. Note that for the far upstream location no 'outer' measurement was conducted due to inability to place sensor due to structural elements.

Numerical setup

In the Large Eddy Simulation, one approximates the solutions of the equations that describe the fluid flow via solving their filtered (coarse grained) form and applying a model for the small scale motion (Sagaut 2006). The governing equations for incompressible, isothermal, non-Newtonian fluid flow under the Large Eddy Simulation (LES) framework are the filtered Navier–Stokes equations, which represent the conservation of mass and momentum. Typically, when applying the filtering to the Navier–Stokes equation for a non-Newtonian fluid, an additional term is introduced, often named τ^{fv} , the so-called 'turbulent viscous stress', due to variation of effective viscosity. Ohta and Miyashita (2014) argue that this term is much smaller than the error associated with the SGS model and it will in this work be disregarded.

The fluid is modelled using a generalized power-law formulation with a yield stress threshold, which resembles a Herschel–Bulkley-type model. This model captures the non-Newtonian behaviour of the fluid, including high resistance to flow below the yield threshold and shear-thinning behaviour at higher shear rates. The apparent viscosity $\tilde{\mu}$ is dependent on the local effective shear rate $\dot{\gamma}$, defined as:

$$\dot{\gamma} = \sqrt{2(\tilde{S}_{ij}\tilde{S}_{ij})}$$

Where $\tilde{S}_{ij}$ is the rate-of-strain tensor for the resolved scale: $\tilde{S}_{ij} = \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right)$ where $\tilde{u}_i$ denotes the resolved velocity in i -th coordinate direction. The viscosity is defined as

$$\tilde{\mu} = \begin{cases} \mu_y, & \text{if } \dot{\gamma} < \dot{\gamma}_y \\ \frac{\tau_y}{\dot{\gamma}} + K \cdot \dot{\gamma}^{n-1} & \text{if } \dot{\gamma} \geq \dot{\gamma}_y \end{cases}$$

Defined for this case as above. To model the effects of unresolved turbulence, the Wall-Adapting Local Eddy-viscosity (WALE) model (Nicoud & Ducros 1999) is employed. WALE offers advantages in wall-bounded flows, as it adapts to both strain and rotation, ensuring accurate near-wall behaviour without the use of damping functions. Non-Newtonian rheology must be coupled with the Navier–Stokes equations to account for variable viscosity. Although the WALE model was originally developed for Newtonian fluids, it can be effectively applied to non-Newtonian flows provided that the interaction with spatially varying viscosity is treated correctly.

The SGS eddy viscosity is calculated as:

$$\mu_{SGS} = \rho(C_w\Delta)^2 \left(\frac{(S_{ij}^d S_{ij}^d)^{\frac{3}{2}}}{(\tilde{S}_{ij}\tilde{S}_{ij})^{\frac{5}{2}} + (S_{ij}^d S_{ij}^d)^{\frac{5}{4}}} \right)$$

where Δ is: $V^{1/3}$, where V is the volume of the cell, C_w equal to 0.544, and S_{ij}^d is a term based on the square of the velocity gradient tensor, see (Nicoud & Ducros 1999).

The numerical setup includes a second-order implicit time integration scheme for unsteady simulations, which provides robust and accurate treatment of transient phenomena required for LES. Convection terms are discretized using a central differencing scheme, preserving resolved turbulent structures without introducing numerical dissipation. The non-Newtonian viscosity is computed dynamically at each time step based on the local resolved strain rate field. The governing equations are solved using the commercial finite volume solver StarCCM+.

The computational geometry is shown in Figure 4. At $19.5 D$ downstream of the inlet is the 90° degree bend (curvature radius $R/D = 1.5$). Downstream of the bend the flow is vertically upwards with the pipe centred at $x = 0$ m. The computational geometry has an outflow at a height of 2.21 m. The upstream part ($10 D$) of the upstream straight pipe section is initially used to recirculate the flow before it enters the remaining part of the geometry including the bend in order to have a fully developed turbulent flow, as explained below. This part of the geometry is indicated in Figure 4 by the vertical dotted line.

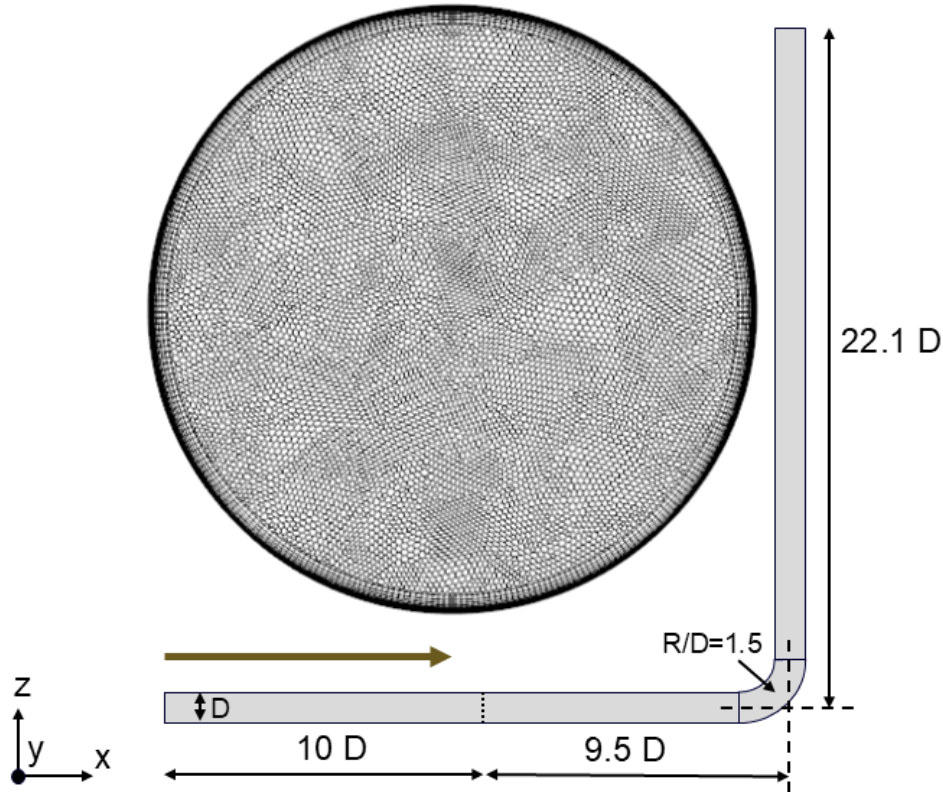


Figure 4: Sketch of the computational domain and a section of the computational grid in the straight section upstream of the 90° degree bend. The dotted line in the upstream straight pipe section indicates the length of the initial straight pipe section where the flow is recirculated in order to attain a fully developed turbulent flow. The horizontal arrow indicates the flow direction.

The computational grid is a directed polyhedral mesh with a prism layer mesh near the boundary, consisting of 20 layers and a stretch factor of 1.3. The full mesh consists of 10.9 million cells of which 3.9 million are in the upstream straight section that is used to attain a fully developed turbulent flow. Of the total number of cells, 6.4 million cells are internal polyhedral cells. The maximum skewness angle is 88.9° and the face validity is good. The wall y^+ values are below 1 (reaching up to 0.33 in the final simulation at maximum) meaning no wall-functions were required.

The simulation of the flow of the domestic slurry in the presented 90° degree bend geometry is a two-step procedure: first the initialisation of the turbulent pipe flow in the upstream straight pipe section followed by the simulation of the flow in the full geometry including the 90° degree bend.

The flow in the upstream straight pipe section is started with the desired mass flow and periodic boundary conditions over this straight pipe section. The inlet of the domain is initialized using the Synthetic Eddy Method (SEM) (Jarrin et al. 2006) with an initial turbulence intensity of 11% to ensure a turbulent flow at the beginning with a correlation length scale of 10% of the hydraulic diameter of the pipe. If the turbulence level and the correlation length scale of the synthetic eddies are sufficiently unstable, the synthetic turbulence will develop in time to fully developed turbulent pipe flow. The initial bulk velocity is similar to the experiment,

1.48 m/s, and the fixed time step size is set such that the Courant number ($u \Delta t / \Delta x$) is below 0.01-0.1 to ensure capturing the transient phenomena within the transition of the SEM initialization towards a fully developed flow and prevent laminarization. The simulations are performed two times with a different fixed time step size; 1e-4 s and 5e-4 s. All velocity profile data is obtained using the small time step size. The pipe has a length of 10 D, that is long enough to capture multiple turbulent length scales over its length. After 5 seconds (7.4 recirculations in the upstream straight pipe section as based on the bulk velocity), the mean wall shear stress is analyzed. The characteristic time scale $t^* (= R/u^*$, where u^* is the friction velocity $u^* = \sqrt{\tau_w/\rho}$), becomes in that case around 0.79 s. The initialization takes 8 s.

After performing the initialization, the periodic flow in the straight pipe is mapped into the domain with the pipe bend at the outflow location of the upstream straight section (10 D) and simultaneously back to the inlet where it further develops. Simulations of around 35 s have been performed to ensure enough time the flow can pass through the pipe and develop. Averaging is done again on approximately the final 7 s where the flow profiles are obtained in a similar way as in the experiment, so almost $10 t^*$. This period for averaging purposes is sufficient (Mason & Callen 1986).

Resulting mean velocity profiles are analyzed at locations: -17 D, -13 D and -1 D upstream of the bend, and at 0 D, 1 D, 4 D, 7 D, 10 D, 13 D and 17.5 D downstream of the bend (all at the three azimuthally oriented radial lines as quantified experimentally).

The minimum of the ratio of resolved turbulent kinetic energy over the subgrid turbulent kinetic energy value found is around 84% and occurs close to the transition region in grid from straight pipe to bend. This agrees with the 'Pope 80% rule' for an accurate LES (Pope 2000, Geurts et al. 2019).

RESULTS

Approach flow

In this section we compare experimental and numerical results of the approach flow in the upstream straight pipe section for average velocity profiles and turbulence.

In Figure 5, results are shown for the wall shear stress from the measurement and the LES, together with different models, namely those described in Chilton and Stainsby (1998), Wilson and Thomas (1985), Dodge and Metzner (1959) and in Slatter (1995). The wall shear stress for pipe flow of water, as determined via the Darcy-Weisbach friction factor using the Colebrook relation (Colebrook & White 1937) is also indicated. In general, the models by Chilton and Stainsby (1998), Wilson and Thomas (1985) lead to results that are comparable to the shear stress for water near the present experimental conditions of $U_b = 1.48$ m/s, but the mentioned models (CS, WT) underpredict the experimental wall shear stress by approximately 0.8 Pa (15% underprediction) and at higher flow rates the underprediction with respect to the experimental values becomes worse. Remarkably, the mentioned models show good agreement with the wall shear stress from the LES. Note that the Slatter (1995) model is special in the respect that at high flow rates this model is closest to the measured values of wall shear stress (but still lower). At the flow rate $U_b = 1.48$ m/s the model by Dodge and Metzner (1959) agrees with the measured wall shear stress up to 0.2 Pa (4%), and the models by Chilton and Stainsby (1998) and Wilson and Thomas (1985) agree better with the wall shear stress from the LES. The wall shear stress of the approach flow in the simulation is around 4.8 Pa, which is approximately 9% lower than the experimental results.

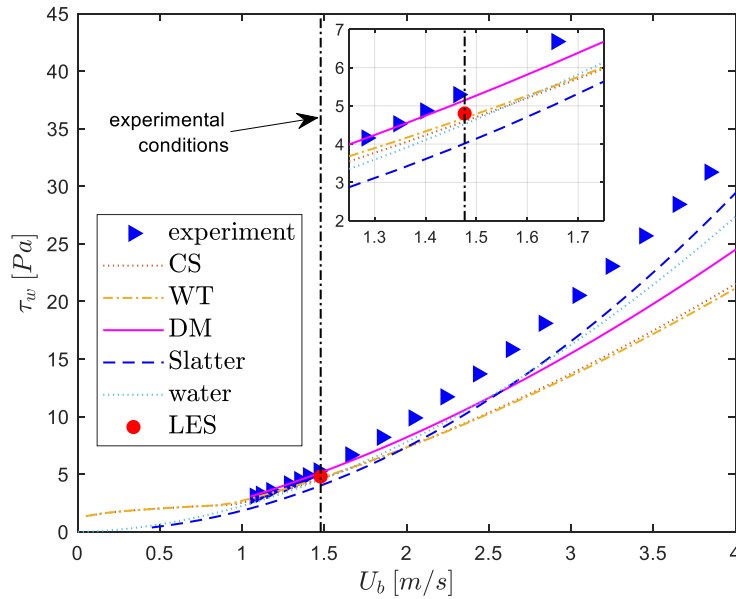


Figure 5: wall shear stress (from pressure drop, from different models, from the LES and for water) versus bulk flow velocity. Inset: zoom near the experimental conditions of interest. CS is Chilton and Stainsby (1998), WT is Wilson and Thomas (1985) and DM is Dodge and Metzner (1959), Slatter is Slatter (1995).

Figure 6 shows average velocity profiles at $x = -17D$ over a full diameter oriented vertically and on a radial line oriented horizontally for experiment and simulation. Agreement up to approximately 3 % difference is found in the bulk region, and remarkably, better agreement in the near wall region. The UIV measurement shows meaningful results virtually up to the pipe radius, where likely the signal to noise ratio becomes too low because of damping and loss of correlation. The results for the average velocity on the profile that is oriented horizontally in the pipe shows comparable results except that the difference near $r/R = 0.35$ is somewhat larger, approximately 5%.

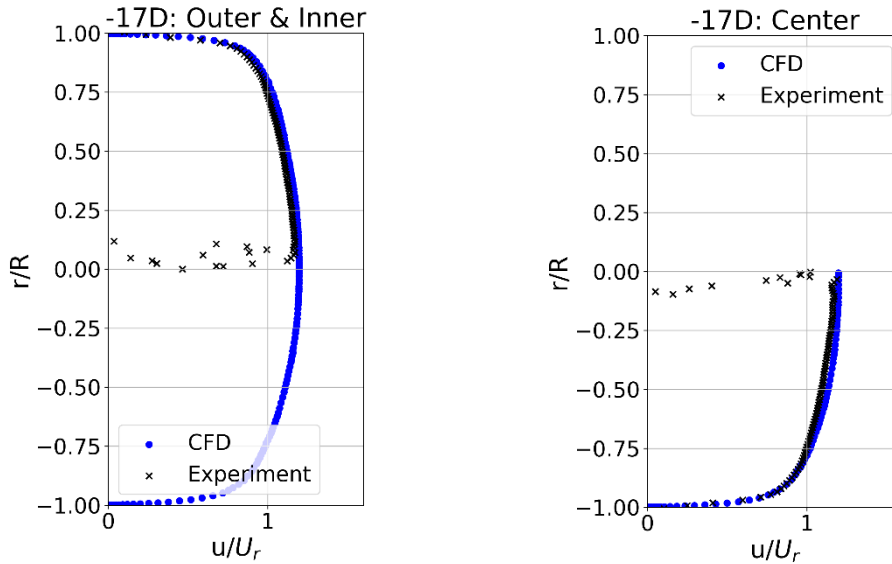


Figure 6: Left: approach flow average velocity profile at $x = -17D$ (experiment and simulation). Left: profiles over a vertically oriented section ('Inner' and 'Outer'). Right: profile on the 'Center' section.

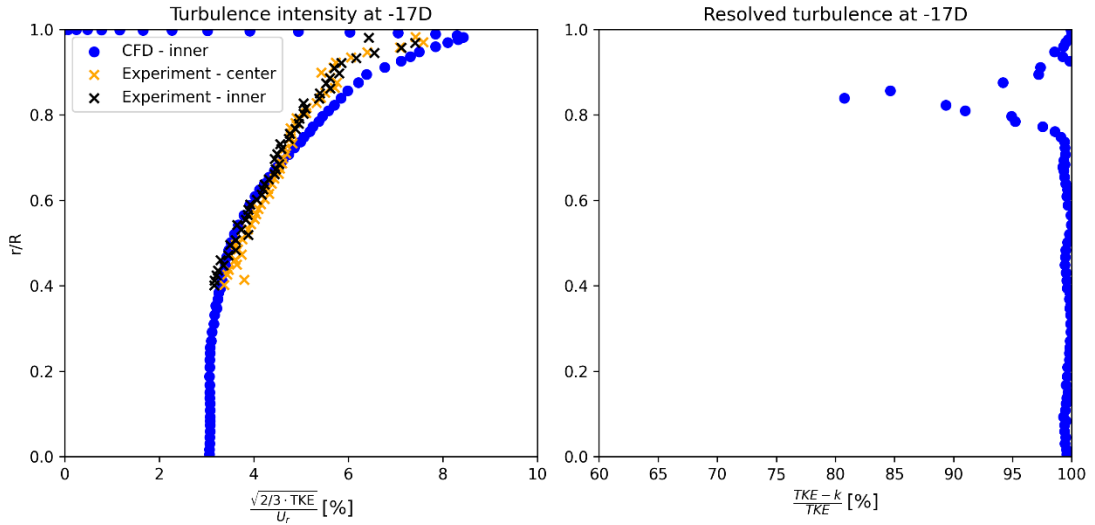


Figure 7: Left: approach flow turbulence intensity profile at $x = -17$ D. (experiment and simulation). Right: approach flow ratio of resolved turbulent kinetic energy over total turbulent kinetic energy.

Figure 7 shows the turbulence intensity profiles for the LES simulation and space-time filtered UIV measurement along with the fraction of resolved turbulent kinetic energy in the simulation. The agreement in turbulence intensity between the simulation and the measurement is reasonably good. The measurement underestimates the turbulence intensity with respect to the simulation in the range from $r = 0.7 R$ to $r = R$ by approximately 10% and mostly overestimates the turbulence intensity in between $r = 0.4 R$ and $r = 0.7 R$ by a small amount. The experimental results show symmetric behaviour. The right panel of Figure 7 shows that the resolved fraction of turbulent kinetic energy is always above 80 % in the approach flow.

All in all it can be concluded that the approach flow is sufficiently well behaved in both the simulation as in the measurement to draw meaningful conclusions about the results near and downstream of the bend.

Flow in and downstream of bend

A visualisation of an instantaneous velocity field from the LES is shown in Figure 8. The velocity map shows that the flow accelerates mainly in the inner section of the bend, up to 2 m/s, causing flow separation and a wake like structure in the inner bend where the curvature ends (at the location of the 0 D profiles). The high velocity 'jet' appears to impinge on the outer wall around +1 D downstream of the end of the curved part of the bend. Remarkably upon close inspection there are local patches of high velocity magnitude central in this wake area. Velocity profiles for inner, outer and center locations 0 D, +1 D, +4 D and +7 D for 'Inner', 'Outer' and 'Center' sections are shown in Figure 9 (grey patches indicate regions where the measurement data has low signal to noise ratio or no signal). At the locations where both reliable measurement data and LES data is available, the difference between the measurement and simulation is approximately 7% at most, where the UIV results are towards lower velocities. No flow reversal is visible. The mentioned phenomena in the 'wake' with a local maximum velocity at $r/R=0.75$ of 0.62 m/s and the main separated jet that impinges on the outer wall are clearly visible in the simulation results, supported by the experimental results outside of the regions of low SNR. At this location of the local maximum intermittent high speed patches are visible in Figure 8 that might be caused by the rotation of the turbulent Dean vortex topology in the cross sectional plane. Figure 9 shows that at location +7 D the profile is not symmetric yet. The instantaneous velocity field in Figure 8 indicates that until at least two-thirds of the downstream length, the flow structure is very different from the upstream flow structure, judging by visual inspection of the spatial velocity variance.

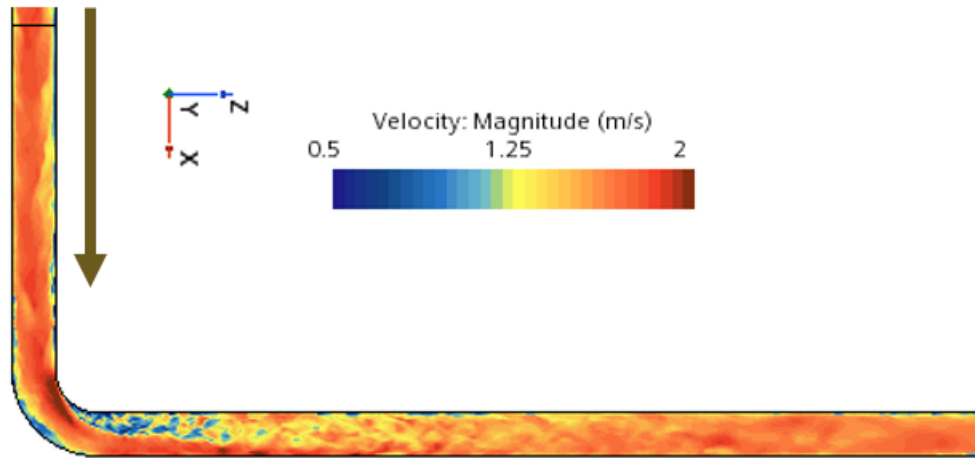


Figure 8: visualisation of instantaneous flow field from the downstream part of the LES domain. Note that the geometry is rotated to fit the page width, and the upstream straight pipe section is not shown. The arrow indicates the flow direction. The color map is clipped for visualisation purpose.

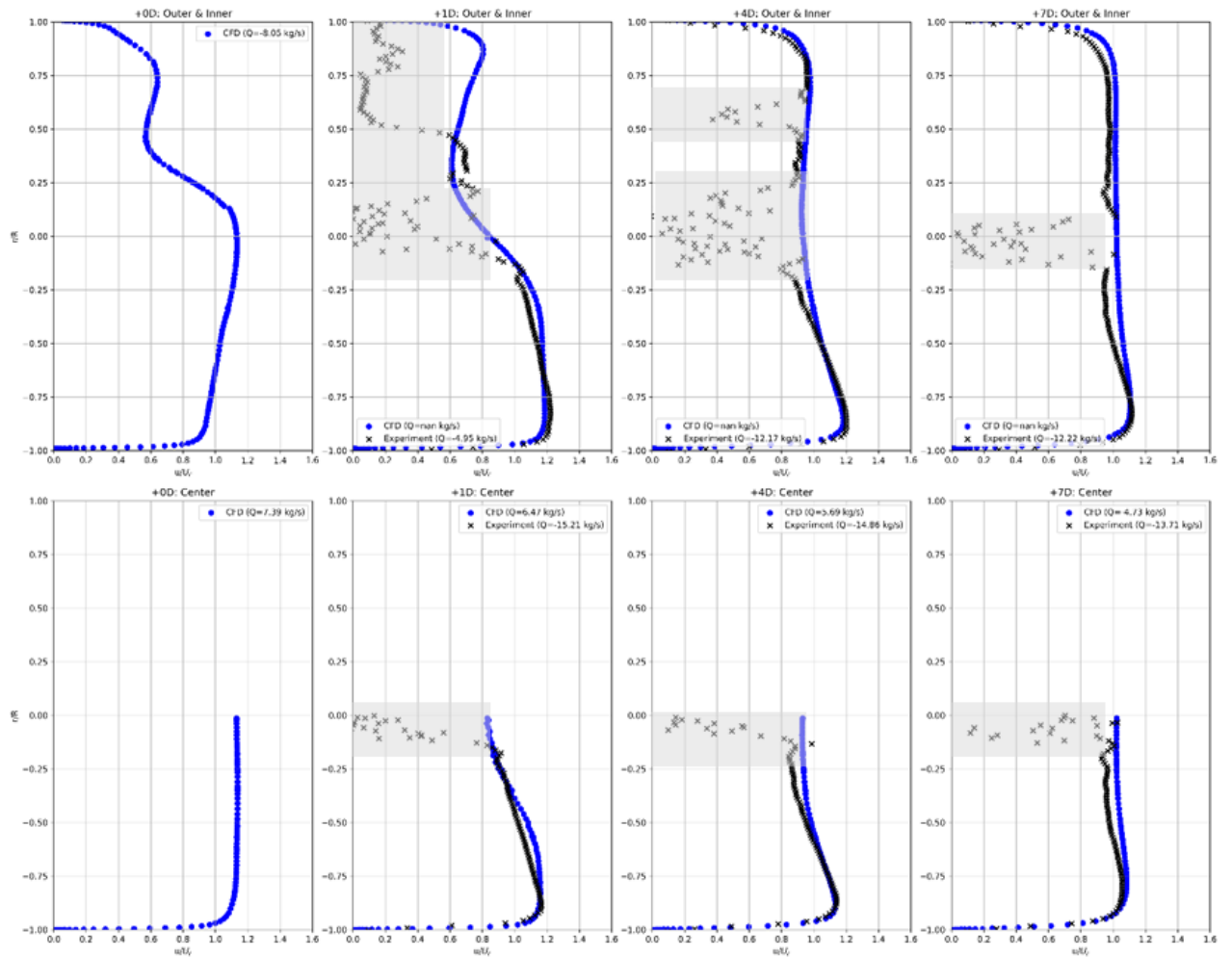


Figure 9: velocity profiles for inner, outer and center locations 0 D, +1 D, +4 D and +7 D for 'Inner', 'Outer' and 'Center' sections. Gray patches indicate areas where the signal to noise ratio is low in the measurement data.

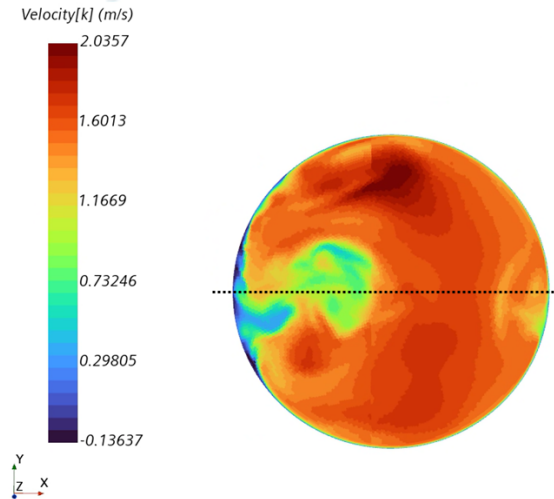


Figure 10: instantaneous velocity field in a cross section at +1 D downstream of the bend. The dotted black line indicates the midplane of the pipe.

Figure 10 shows an instantaneous velocity field in a cross section at +1 D downstream of the bend. The velocity field resembles an asymmetric Dean vortex topology (Dean 1927) as is expected for high inertia flow in a bend (Kalpakli Vester et al. (2016)). The (quasi-)periodic rotation in the indicated cross section of the instantaneous turbulent Dean-like vortices possibly leads to the intermittent high speed patches as observed in Figure 8 and the local maximum velocity as observed in Figure 9. Averaged velocity contours downstream of the bend are shown in Figure 11 along with streamlines of the average flow at locations +0 D, +1 D, +4 D, +7 D as in the LES. The behaviour along the flow direction can be seen to indeed form a wake structure. It is observed that at +7 D the two Dean-type vortices that are characteristic for a bend are still observable in the streamline pattern.

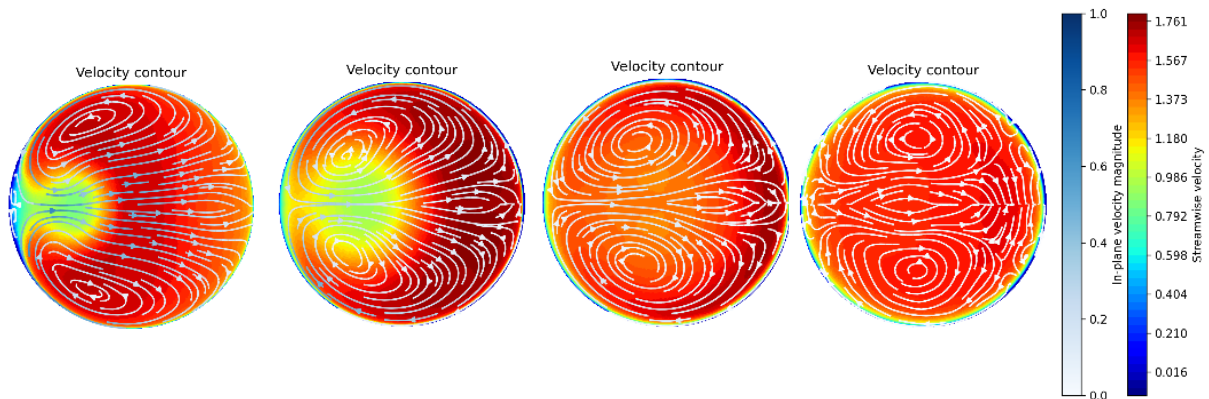


Figure 11: Mean velocity profiles through cross section after bend over around 14 s showing the development of Dean-type vortices. From left to right: +0 D, +1 D, +4 D, +7 D.

Figure 12 shows velocity profiles for inner, outer and center locations +10 D, +13 D and +17.5 D for 'Inner', 'Outer' and 'Center' sections for measurement and simulation. At +10 D the asymmetry in the flow is still clearly visible via the somewhat higher velocity in the 'outer' bend than in the 'inner' bend. At +17 D the relevant results are subtle but important. At 17.5 D the asymmetry is present in the experimental data, most clearly in the 'outer' and 'inner' profile. This is not, or much less, the case for the LES results.

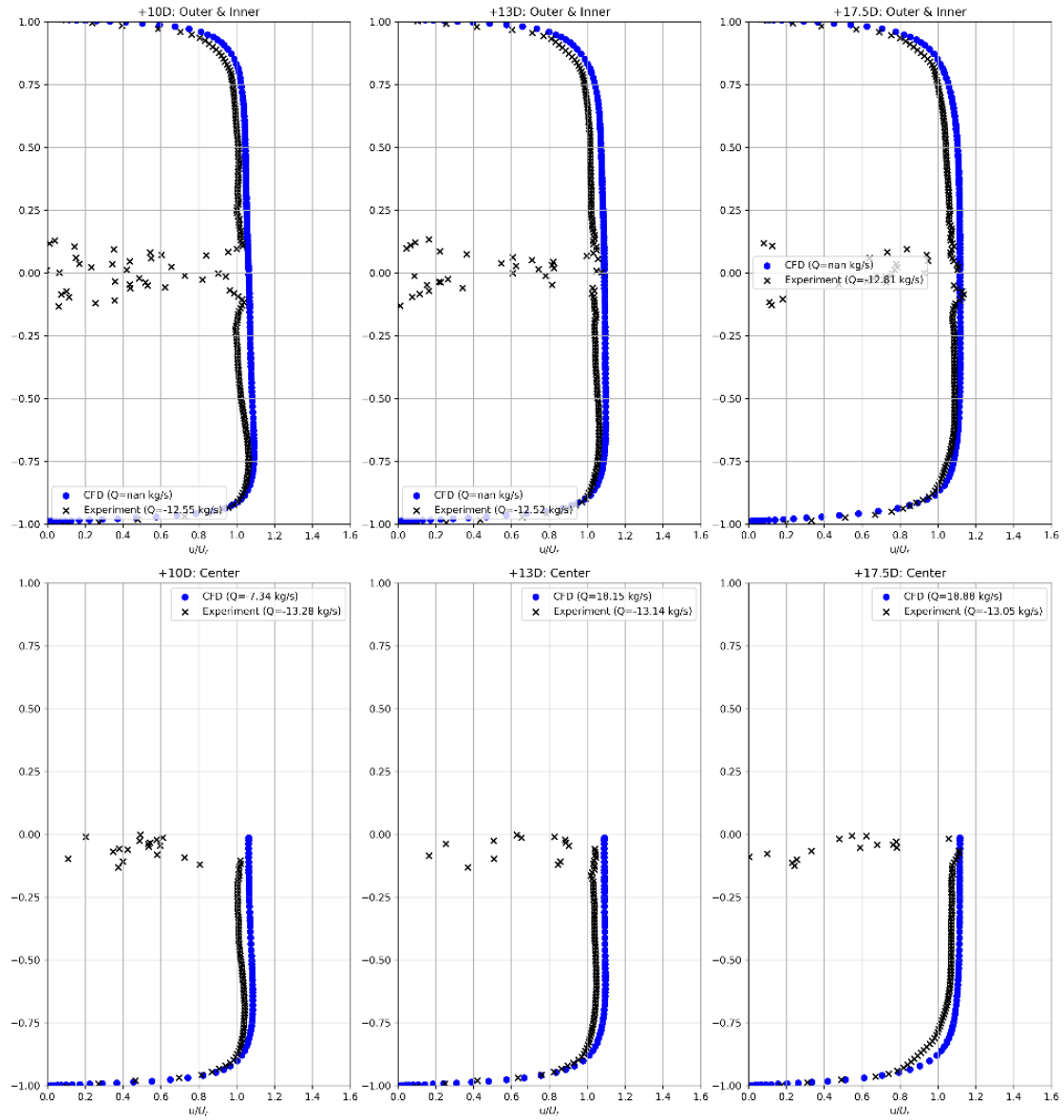


Figure 12: velocity profiles for inner, outer and center locations +10 D, +13 D and +17.5 D for 'Inner', 'Outer' and 'Center' sections for measurement and simulation.

DISCUSSION

First, the application of the selected UIV and LES methods for pipe flow of non-Newtonian fluids will be discussed , and afterwards the flow physics is discussed.

UIV method for non-Newtonian flows for pipeline hydraulics

The application of the present form of the UIV method to quantify the flow through a 90 degree bend yields useful velocity data, over a radial range of up to half the diameter per sensor location for the present fluid and flow facility, including TKE in the straight approach flow. The flow in the inner section of the 90 degree bend is likely too 3-dimensional to obtain reliable mean velocity profiles.

LES method for non-Newtonian flows for pipeline hydraulics

The application of the present implementation of the LES method to quantify the complex dynamics of the flow through a 90 degree bend yields useful data for pipeline hydraulics applications, especially in combination with experimental data, where data is obtained in experimentally difficult to access locations such as the inner bend. One observation is that the turbulence level decreases during the simulation, which might eventually result in not maintaining a turbulent state. Note that relaminarization has not been observed in the present work. This has been observed before for non-Newtonian fluids with a power law index as low as $n = 0.5$, see Inagaki (2021).

Approach flow

Upon comparison of the approach flow between the measurements and simulations we find that the wall shear stress is approximately 9% higher in the experiments than in the LES simulation. Agreement with different known analytical models differs, where the model by Dodge and Metzner (1959) favors the experimental data and the models by Chilton and Stainsby (1998) and Wilson and Thomas (1985) favor the LES data.

Agreement in average velocity profiles is in the order of 5% of U_b and small (10% max) differences in turbulence intensity, where the UIV measurement results indicate lower velocities than the simulation. Regarding the profiles this agrees with the known bias towards lower velocities in PIV methods, see (Adrian and Westerweel 2011). The fraction of resolved over total turbulent kinetic energy in the LES is over 0.8 so this is sufficient to be an 'accurate LES' (Pope 2000).

Because of these apparently contradictory results, we will assume that the difference in τ_w between experiment and simulation is related to the following issues:

- inaccuracy in the LES model (disregard of the turbulent viscous stress τ^{fv} and/or accuracy in the SGS model).
- the fact that the rheological characterization is performed in a classical manner (via a simple shear flow in a rheometer), in which the flow condition is very different from turbulent pipe flow, particularly the absence of elongation (see Nieuwstadt and Den Toonder 2001).

Flow of domestic slurry in 90 degree bend

For the flow in the 90 degree bend we find that, at locations where both reliable UIV velocity data and LES data is available, the maximal difference between experimental and model results for the average velocity is 7%, where the ultrasound data shows systematically lower values than the LES data. The ultrasound measurement has difficulties to attain a sufficient signal to noise ratio for the 'inner region' of the bend, possibly due to the quasi-periodically fluctuating turbulent Dean vortices that are present in the LES results; the out of plane loss of particle pairs and corresponding loss of correlation is suspected to be the cause for this. An additional challenge is the connectivity between the straight ultrasound probe and the rigid curved bend geometry.

Flow recovery

Interestingly, we find that, for the experimental data, the velocity profiles are not yet recovered fully at 17.5 D downstream of the bend, but for the LES data the average velocity profile is virtually symmetric, indicating that care must be taken when interpreting LES data in terms of flow profile recovery after a bend for a water-clay mixture that is modelled in the simulation using a Herschel Bulkley model. An additional implication of this difficulty in accurately modelling the dissipation is mentioned below.

In, for example Figure 10 in Kalpakli Vester et al. (2016) it is shown that the full pressure loss for *water* in a bend is only recovered fully at approximately 35 D downstream of the bend (although this bend has a different curvature ($R/D = 1.85$)). Laser doppler anemometry results from Straka et al. (2020) indicate that for water in a 90 degree bend with $R/D = 1.425$ the velocity profile is not yet recovered at a distance of 30 D downstream of the bend at $Re = 50000$ and $U_b = 0.400$ m/s.

Additional finding

An additional finding on the LES is that the turbulence intensity initialisation value is relatively high as compared to water (11%) after which a steady turbulent flow develops. It is observed that the steady turbulent flow slowly but surely decreases in turbulence intensity. This implies that the present LES simulation, despite using a central differencing scheme, has difficulty in correctly modelling the dissipation, as observed by for example Qi et al (2022).

CONCLUSIONS AND RECOMMENDATIONS

In this paper, we compare measurements and simulations of the flow of a non-Newtonian fluid with a finite yield stress to obtain insights in the performance of simulations as a tool to determine different hydraulic and hydrodynamic parameters, such as: the general characteristics of the flow, the performance of the combined LES model with WALE SGS model and rheological modelling for this complex fluid, and the recovery of the velocity profile downstream of the bend.

The main conclusions are that:

1. Despite downsides such as weak signal combined with a turbulent wake region in the inner bend for the UIV measurements, and the errors associated with the SGS scale model and modelling assumptions in the LES, the combination of UIV measurements and LES simulations of the considered non-Newtonian flow lead to a relatively clear characterization of the flow in and around a 90 degree bend, as evaluated via a comparison of the measured and simulated velocity profiles and the expected uncertainties in both. The shortcomings of the combination of UIV and the present implementation of the LES methods fall within engineering accuracy.
2. A small difference in symmetry of the velocity profile near the outflow boundary, 17.5 D downstream of the bend between the measurement and the simulation, indicates that the simulation of the homogenous modelled water-clay mixture recovers too close to the bend. This is likely a symptom of the difficulty the LES model with WALE SGS model has with correctly modelling the dissipation.

The main recommendations for future work are to:

1. perform a comparison of the pressure loss due to the bend between the UIV measurement and the LES simulation, and compare the result to the pressure loss for water (from one or multiple standard hydraulics handbooks and possibly from measurements) to get insights on the behavior of non-Newtonian fluids with a finite yield stress.

LITERATURE

1. Chilton, R.A and Stainsby R., 1998, Pressure loss equations for laminar and turbulent non-Newtonian pipe flow, *J. Hydraulic Eng.*, vol.124, no 5, pp522-529.
2. Wilson, K.C., and A.D. Thomas., 1985, A new analysis of the turbulent flow of non-Newtonian Fluids, *Canadian J. Chemical Engineering*, vol 63, pp.539-546.
3. Dodge, D. W., & Metzner, A. B. (1959). Turbulent flow of non-Newtonian systems. *AIChE journal*, 5(2), 189-204.
4. Slatter, P.T. (1995). Transitional and turbulent flow of non-Newtonian slurries in pipes (PhD thesis, University of Cape Town, South Africa).
5. Dash, A., & Poelma, C. (2022). Long-time-scale transients in an industrial-scale slurry pipeline near the laminar–turbulent transition. *Flow*, 2, E25.
6. Dash, A., Hogendoorn, W., Oldenziel, G., & Poelma, C. (2022). Ultrasound imaging velocimetry in particle-laden flows: Counteracting attenuation with correlation averaging. *Experiments in Fluids*, 63(3), 56.
7. Thota Radhakrishnan, A., Poelma, C., Van Lier, J., & Clemens, F. (2021). Laminar-turbulent transition of a non-Newtonian fluid flow. *Journal of Hydraulic Research*, 59(2), 235-249.
8. Thota Radhakrishnan, A. (2019). Domestic slurry hydraulics in transport (Doctoral dissertation, Doctoral thesis, Delft University of Technology).
9. Mehta, D., Thota Radhakrishnan, A. K., van Lier, J. B., & Clemens, F. H. (2021). Assessment of numerical methods for estimating the wall shear stress in turbulent Herschel–Bulkley slurries in circular pipes. *Journal of hydraulic research*, 59(2), 196-213.
10. Mehta, D., Thota Radhakrishnan, A. K., Van Lier, J., & Clemens, F. (2018). A wall boundary condition for the simulation of a turbulent non-Newtonian domestic slurry in pipes. *Water*, 10(2), 124.
11. Yusufi, B. K., Kapelan, Z., & Mehta, D. (2024). Rheology-based wall function approach for wall-bounded turbulent flows of Herschel–Bulkley fluids. *Physics of Fluids*, 36(2).
12. Wang, K., Sprinkle, B., Zuo, M., & Ristroph, L. (2024). Centrifugal flows drive reverse rotation of Feynman’s sprinkler. *Physical Review Letters*, 132(4), 044003.
13. Kalpakli Vester, A., Örlü, R., & Alfredsson, P. H. (2016). Turbulent flows in curved pipes: Recent advances in experiments and simulations. *Applied Mechanics Reviews*, 68(5), 050802.
14. Dash, A. (2022). Opaque inertial suspensions. (Doctoral dissertation, Doctoral thesis, Delft University of Technology).
15. Hogendoorn, W., & Poelma, C. (2018). Particle-laden pipe flows at high volume fractions show transition without puffs. *Physical review letters*, 121(19), 194501.
16. Radhakrishnan, A. T., Van Lier, J. B., & Clemens, F. H. L. R. (2018). Rheological characterisation of concentrated domestic slurry. *Water research*, 141, 235-250.
17. A. Talmon (2023), segregating non-Newtonian slurries, Lecture notes.
18. Idelchik, I. E., Steinberg, M. O., & Martynenko, O. G. (1986). *Handbook of hydraulic resistance* (Vol. 2). New York: Hemisphere publishing corporation.
19. Brücker, C. (1998, September). A time-recording DPIV-study of the swirl switching effect in a 90° bend flow. In *Proceedings of the Eight International Symposium on Flow Visualization, Sorrento, Italy* (Vol. 20, pp. 21-22).
20. Hufnagel, L., Canton, J., Örlü, R., Marin, O., Merzari, E., & Schlatter, P. (2018). The three-dimensional structure of swirl-switching in bent pipe flow. *Journal of Fluid Mechanics*, 835, 86-101.
21. Poelma, C. (2017). Ultrasound imaging velocimetry: a review. *Experiments in Fluids*, 58(1), 3.
22. Meneveau, C., & Katz, J. (2000). Scale-invariance and turbulence models for large-eddy simulation. *Annual Review of Fluid Mechanics*, 32(1), 1-32.
23. Krishnan Thota Radhakrishnan, A., Poelma, C., Van Lier, J., & Clemens, F. (2021). Laminar-turbulent transition of a non-Newtonian fluid flow. *Journal of Hydraulic Research*, 59(2), 235-249.

24. Colebrook, C. F., & White, C. M. (1937). Experiments with fluid friction in roughened pipes. *Proceedings of the Royal Society of London. Series A-Mathematical and Physical Sciences*, 161(906), 367-381.
25. Adrian, R. J., & Westerweel, J. (2011). *Particle image velocimetry* (No. 30). Cambridge university press.
26. Zhou, B., Fraser, K. H., Poelma, C., Mari, J. M., Eckersley, R. J., Weinberg, P. D., & Tang, M. X. (2013). Ultrasound imaging velocimetry: Effect of beam sweeping on velocity estimation. *Ultrasound in Medicine & Biology*, 39(9), 1672-1681.
27. Sagaut, P. (2006). *Large eddy simulation for incompressible flows: an introduction*. Berlin, Heidelberg: Springer Berlin Heidelberg.
28. Nicoud, F., & Ducros, F. (1999). Subgrid-scale stress modelling based on the square of the velocity gradient tensor. *Flow, turbulence and Combustion*, 62(3), 183-200.
29. Mason, P. J., & Callen, N. S. (1986). On the magnitude of the subgrid-scale eddy coefficient in large-eddy simulations of turbulent channel flow. *Journal of Fluid Mechanics*, 162, 439-462.
30. Jarrin, N., Benhamadouche, S., Laurence, D., & Prosser, R. (2006). A synthetic-eddy-method for generating inflow conditions for large-eddy simulations. *International Journal of Heat and Fluid Flow*, 27(4), 585-593.
31. Geurts, B. J., Rouhi, A., & Piomelli, U. (2019). Recent progress on reliability assessment of large-eddy simulation. *Journal of Fluids and Structures*, 91, 102615.
32. Pope, S. B. (2000). *Turbulent Flows*. Cambridge University Press, 2000.
33. Dean, W. R. (1927). XVI. Note on the motion of fluid in a curved pipe. *The London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science*, 4(20), 208-223.
34. Tunstall, M. J., & Harvey, J. K. (1968). On the effect of a sharp bend in a fully developed turbulent pipe-flow. *Journal of Fluid Mechanics*, 34(3), 595-608.
35. Straka, M., Koglin, C., & Eichler, T. (2020). Segmental orifice plates and the emulation of the 90-bend. *tm-Technisches Messen*, 87(1), 18-31.
36. Adrian, R. J., & Westerweel, J. (2011). *Particle image velocimetry* (No. 30). Cambridge university press.
37. Ohta, T., & Miyashita, M. (2014). DNS and LES with an extended Smagorinsky model for wall turbulence in non-Newtonian viscous fluids. *Journal of Non-Newtonian Fluid Mechanics*, 206, 29-39.
38. Inagaki, M. (2021). Large eddy simulation of non-Newtonian viscous fluids with low grid dependency using an anisotropy-resolving subgrid-scale model. *Journal of Non-Newtonian Fluid Mechanics*, 295, 104603.
39. Nieuwstadt, F. T. M., & Den Toonder, J. M. J. (2001). Drag reduction by additives: a review. *Turbulence structure and modulation*, 269-316.
40. Qi, H., Li, X., & Yu, C. (2022). A modified wall-adapting local eddy-viscosity model for large-eddy simulation of compressible wall-bounded flow. *Physics of Fluids*, 34(11).