Experimental and numerical investigation of the flow field inside a dense slurry mixer

Booklet of the PhD Dissertation

Written by: Péter Csizmadia
M.Sc. in Mechanical Engineering

Supervisor: Dr. Csaba Hős
associate professor

1st September 2016, Budapest
Chapter 1

Introduction

The aim of the present thesis is to study the flow in an industrial dense slurry mixer. The sketch of the device can be seen in Figure 2. For the scientific investigations the device was divided into parts, represented by the three chapters of the dissertation. First, straight-pipe friction loss and loss coefficients of elbows and diffusers were examined (Chapter 2). Then, the cyclone-like flow field is described inside the mixer (Chapter 3). Finally, an experimental investigation of jet breakup length of coal ash slurries in air is presented (Chapter 4). Moreover, the effect of the rheological behaviour of the used fluids has also been tested.

The pressure losses of straight pipes and components are basically well-known in the case of Newtonian fluids but few data are available when the rheological properties are non-Newtonian. The used CFD models for predicting the loss coefficients of elbows and diffusers have been validated with my own experiments for Newtonian (water) as well as for non-Newtonian (power-law) fluids. For Bingham plastic materials, critical Hedström-numbers and the loss coefficients have been determined. The loss coefficients are given explicitly and increase dramatically if the actual Hedström-numbers are above the critical values. This chapter reports on the experiences of the "as-is" set-up of the ANSYS CFD code for using it in the case of power-law and Bingham plastic fluids.

The acquired knowledge is applied in the study of the cyclone-like flow field inside the hydrodynamic mixer too, although the used CFD models have also been validated with my own LDV measurements. According to the results, in the case of water, a 7-equation turbulence model (BSL) and unsteady computations are needed to describe the flow field. In contrast, the 2-equation turbulence model (SST) and stationary simulations provide good results with respect to the examined power-law fluid. Furthermore, the tangential flow field displays a hyperbolic behaviour only in the case of water.

Finally, an experimental study was carried out in order to determine the jet breakup length of non-Newtonian, Bingham plastic slurries in air, notably to obtain an experimentally verified formula for the breakup length. The ash-to-water concentration varies between appr. 9% and 57%. Five dimensionless formulae can have influence on the process from which the Froude- and the Weber-numbers dominate. With the help of the method of least squares, explicit equations are given to estimate the dimensionless jet breakup length.
Chapter 2

CFD-based estimation and experiments on the loss coefficient for Newtonian and non-Newtonian fluids through diffusers and elbows

2.1 Introduction

The importance of non-Newtonian fluids in engineering applications is continuously increasing as these materials are often encountered in e.g. energy, petroleum or food industry. Due to their complex rheological behaviour standard fluid mechanical techniques are often not applicable or have to be modified: on the one hand, the definition of such basic quantities as e.g. the Reynolds number is highly nontrivial for a power-law fluid or, on the other hand, for a Bingham plastic fluid, new dimensionless properties appear such as the Hedström number. However, when designing pipeline systems for such fluids, one has to estimate the pressure losses to be able to choose the appropriate pump and pipe diameters, which is often cumbersome due to the lack of experimental and theoretical guidelines. Computational fluid dynamics (CFD) codes seem to be suitable for such computations but there is little experience on the behaviour of such codes (notably the performance of the turbulence models developed for Newtonian fluids) in the case of non-Newtonian fluids. This chapter reports on the experiences of the "as-is" application of a commercial CFD code for predicting loss coefficients in the case of some non-Newtonian fluids, i.e. I did not modify or fine-tune the standard settings of the code ANSYS CFX®.

Several experimental and numerical studies can be found in the literature on the pressure drop in pipelines and fittings due to friction losses and shape effects for non-Newtonian fluids. [1] reports friction losses in valves and fittings, including seat, ball, no-return, and butterfly valves, bends, tees, and unions. The fluids in this study were food products, which behave like power-law fluids. [2] examines the pressure drop in ice slurry flow and compared different rheological models, as the Bingham, Casson, power-law and Herschel-
Bulkley models. This paper contains the definition of a generalized Reynolds number $Re_{PL}$. The behaviour of the same slurry in horizontal pipes is also examined in [3].

After the literature overview it can be seen that there is a large body of experimental (and analytical) studies on non-Newtonian loss coefficients. But the penetration of CFD codes is rather weak. The results of the present study encourages the use of these codes for such purposes.

The definition of the loss coefficient

The loss coefficient is defined as the non-dimensional difference in total pressure between the two ends of a pipe or fitting (see [4], [5]): \[ \zeta = \Delta p \left( \frac{\rho v_{\text{in}}^2}{2} \right)^{-1}, \]

in which $\Delta p$ is the total pressure drop across the element, $\rho$ is the fluid density and $v_{\text{in}}$ is the average velocity at the upstream side. In the case of laboratory experiments or CFD computations one usually does not measure the pressure directly at the upstream and downstream side of the actual element but straight pipe segments are attached to both sides.

Rheology and dimensionless parameters

The rheological law can be written in the following form: $\tau = \mu \dot{\gamma}$ (see e.g. [6]), where $\tau$ is the shear stress, $\dot{\gamma}$ is the shear rate, and $\mu$ is the dynamic viscosity in the case of the non-Newtonian fluids. In the thesis, two non-Newtonian fluids were investigated: Bingham plastic fluids and a power-law material.
Power-law fluid: Carbopol solution

We introduce the material properties of the Carbopol solution, which contains 0.13 m/m\% Carbopol 971, 0.05 m/m\% NaOH and 99.82 m/m\% water. The measurement of the rheology was performed on a Rheotest 2 RV2 rotational viscometer. The results of the rheological measurements and the curve fit of the shear stress - shear rate relationship is described by $\tau = \mu_{PL} \dot{\gamma}^{n_{PL}}$, with $\mu_{PL}$ being the consistency index and $n_{PL}$ standing for the flow behaviour index, see e.g. [2] for details. The actual values were found to be $\mu_{PL} = 0.1334$ Pas$^{n_{PL}}$ and $n_{PL} = 0.7266$.

Bingham plastic material

In the case of the Bingham plastic fluids, the rheological behaviour is described by $\tau = \tau_{yield} + \mu_{B} \dot{\gamma}$ with $\mu_{B}$ viscosity consistency and $\tau_{yield}$ yield stress, see [2]. In the case of Bingham fluids, the dimensionless number highlighting the importance of yield stress is the Hedström number, which is defined as $He = D^2 \tau_{yield} / \rho_{B} \mu_{B}^2$, with $D$ standing for the pipe diameter and $\rho$ for density, see [3].

2.2 CFD modeling

The CFD computations were performed with the commercial code ANSYS CFX© and ICEM CFD was employed as a meshing tool. The software solves the Reynolds-averaged Navier-Stokes equations, the continuity equation (see e.g. [7, 8, 9, 10]), and the transport equations associated with the actual turbulence model. Both eddy-viscosity models ($k-\epsilon$, SST) and Reynolds stress model (BSL Reynolds stress) were tested. Fully structured 3D mesh was used, which was optimized via a grid-independence study: three different meshes were tested in the case of a the $R/D = 1$ elbow. Additional straight pipes were added to the upstream (50D) and downstream (10D length) sides in both cases (elbow and diffuser) to allow proper boundary conditions (see [11]). Fully developed turbulent velocity distribution was prescribed at the inlet and average static pressure at the outlet. The rest of the surfaces were set to no-slip walls. Steady state computations were performed, similar to the ones in [11]. High-resolution spatial scheme was used for all equation classes.

2.3 Results

Although the loss coefficients were determined for elbows and diffusers in the case of both Carbopol solution and Bingham plastic fluids, the Theses #1 and #2 contain results just for Bingham plastic materials.

Figure 2.2 shows the loss coefficients in the case of an elbow of relative radius $R/D = 1$ (above) and diffusers for $A_2/A_1 = 2.25$ (below), for Bingham plastic fluids. In both cases, the critical Hedström-numbers and the loss coefficients have been determined. The loss
coefficients are given explicitly and increase dramatically if the actual Hedström-numbers are above the critical values.

**Thesis #1**

The numerically determined loss coefficient of elbows of relative radii $1 \leq R/D \leq 10$ for Bingham plastic fluids is constant to a $He_{crit}$ critical Hedström number and it increases as a power-law function in the range of $Re_B = 178.6 \ldots 17860$ Reynolds number. The actual values are given:

$$
\zeta_{\text{elbow}} = \begin{cases} 
\zeta_{0,K} & \text{if } 1 < He < He_{crit} \\
 a_K \cdot H e_{crit} & \text{if } He_{crit} < He < 10^8, 
\end{cases} \quad (2.1)
$$

where

$$
\zeta_{0,K}(Re_B) = 40.77 Re_B^{-0.4882},
$$

$$
He_{crit}(Re_B) = a_1 \left( \frac{R}{D} \right)^{b_1}, \quad a_1(Re_B) = 0.6136 Re_B^{1.628},
$$

$$
b_1(Re_B) = -7.108 \cdot 10^{-2} \cdot \ln(Re_B) - 0.1275,
$$

$$
a_K(Re_B) = a_2 \frac{R}{D} + b_2, \quad a_2(Re_B) = 31.36 Re_B^{-2.062}, \quad b_2(Re_B) = -0.0893 Re_B^{-1.708},
$$

$$
b_K = 0.9854.
$$

The coefficients of determination of the curve fits are $R^2 = 0.983, 0.906, 0.999, 0.996, 0.998; 1, 0.953$, respectively.

**Thesis #2**

The numerically determined loss coefficient of diffusers with cross-section ratios $A_2/A_1 = 2.25 \ldots 4$ and $\delta = 7.5^\circ \ldots 40^\circ$ bevel-angel for Bingham plastic fluids is constant up to the $He_{crit}$ critical Hedström number and it increases as a power-law function in the range of $Re_B = 178.6 \ldots 17860$ Reynolds number. The actual values are given:

$$
\zeta_{\text{diffuser}} = \begin{cases} 
\zeta_{0,D} & \text{if } 1 < He < He_{crit} \\
 a_D \cdot H e_{crit} & \text{if } He_{crit} < He < 10^8, 
\end{cases} \quad (2.2)
$$
where

\[ \zeta_{0,D}(Re_B) = 2.878 Re_B^{-0.242}, \]

\[ He_{crit}(Re_B) = 0.01963 Re_B^{1.80}, \]

\[ a_D(Re_B) = 227.9 Re_B^{-2.09}, \]

\[ b_D = 0.9863. \]

The coefficients of determination of the curve fits are \( R^2 = 0.824, 0.996, 0.997 \), respectively.

Publications for Theses #1 and #2: T1, T3, T5.
Figure 2.2: The numerically determined loss coefficients of elbows (above) and diffuser ($A_2/A_1 = 2.25$) (below) in the case of Bingham plastic fluids at different Hedström numbers. a: $Re_B = 178.6$, b: $Re_B = 1786$, c: $Re_B = 17860$. 
Chapter 3

Flow field inside the mixer

3.1 Introduction

The actual industrial application is a dense slurry mixer which is used in coal-fired power plants where dry bed ash and water are mixed and the mixture is pumped to a deposition site. Computational Fluid Dynamics (CFD) techniques play a central role in the improvement of these mixers but experimental results are also extremely important to validate the CFD approach. Thus, a scaled-down mixer was built with Plexiglas walls allowing visual access to the flow for the LDV system. The original fluid (slurry) is a non-Newtonian material which is not suitable for the LDV measurements because of visual problems. Hence another non-Newtonian fluid — Carbopol solution — was chosen which allows visual access up to a limited radial depth.

Due to the highly swirling flow field inside the mixer, capturing the velocity profiles by CFD is a challenging task, not to mention the additional complexity due to the non-Newtonian fluid. Hence, the results of the current study are intended to serve as validation cases for the CFD simulation of both Newtonian and non-Newtonian fluids. Moreover, the presented results also contribute to the understanding of the flow field inside similar mixers, which are nowadays designed mostly based on anecdotes and on-site experiences.

Several researchers deal with the determination of the flow field inside hydro-cyclones which are used not only for separating solid contamination (particles of different sizes) but also for mixing purposes (e.g. static mixers). Some studies apply numerical methods, for instance [12], in which the authors compare three turbulence models and apply their results to improve the geometry of the hydro-cyclones in [13]. Other studies concentrate on predicting separation efficiency by means of CFD (e.g. [14]) and predicting the effect of varying geometries (e.g.: [15]). [16] studies a water and diesel fuel separator numerically for several geometries in which the separation efficiency can improve. In [17], solid-liquid suspension was investigated with the CFD technique and their results show a fairly good agreement with the literature. In [18] solid-liquid separation processes were tested experimentally. [19] applies both numerical and experimental techniques for studying the separation inside a hydro-cyclone. In [20] four different turbulence models were tested and
compared against experimental results and the best agreement was found using the RSM model.

3.2 Experimental set-up

In the mixer the fluid motion is driven by two tangential jets fed by a centrifugal pump. A control valve is mounted to set and measure the recirculated flow rate with the help of the pressure taps measuring the pressure drop on the valve. The axial and circumferential velocity components were measured by the two-component TSI TLN06-363 LDV system (1) which was positioned by a traverse system (2) in three dimensions. The examined fluids were water and the previously mentioned Carbopol solution.

3.3 CFD set-up

A fully structured grid was built with the help of ICEM CFD®, consisting of 1.7M elements, this spatial resolution was found to be suitable to obtain grid-independent solutions. ANSYS CFX® was used as flow solver. The examined geometries can be seen in Figure 3.1, the ratios of the height and the diameter of the mixer are $H/D = 1, 1.391, 2$. Although it is well-known from the literature that the flow is inherently unsteady, both steady-state and transient computations were performed, not only to reduce the computational time but also to gain some experiences on the quality of such solutions. Single-phase (water and Carbopol solution) models was employed with $k-\varepsilon$, SST, and BSL turbulence models. Simple inlet boundary condition with prescribed mass flow rate was set at the tangential jets and outlet at the pressure side of the pump, with prescribed pressure. Convergence was judged by monitoring the velocity at several locations in the flow field and the computation was stopped once all these components reached a quasi-steady state.

Figure 3.1: The sketch of the numerical models, $H/D = 1, 1.391, 2$. 

9
3.4 Results

Although the flow field inside the mixer were determined for water too, Theses #3 and #4 contains results just for the Carbopol solution.

Figure 3.2: Dimensionless tangential (left panels) and axial (right panels) velocity profiles in the case of fully filled tank at different axial positions. The fluid is Carbopol solution, the flow rate is the maximum value, and $H/D=1.391$.

Thesis #3

The cyclone-like flow field was examined in the case of a non-Newtonian, power-law material (Carbopol solution), which contains Carbopol dust (0.13%), NaOH (0.05%), and water (99.82% by weight). The rheological law of the fluid can be described by $\tau = \mu_{PL}\dot{\gamma}^{n_{PL}}$, where $\tau$ is the shear stress, $\dot{\gamma}$ is the shear rate, $\mu_{PL} = 0.1334 \text{Pas}^{n_{PL}}$ is the dynamic viscosity, and $n_{PL} = 0.7266$ is the flow behaviour index. The mixer consists of two, tangential fluid jets, which drive the rotating motion of the fluid body. The distance between the top of the mixer and the jets is $0.184D$ ($D = 0.435 \text{ m}$), and the ratio of the height and diameter of the tank is $1 \leq H/D \leq 2$.

In this case, the 2-equation turbulence model (SST) and stationary simulations provide good results respect to the examined power-law fluid.

Thesis #4

The cyclone-like flow field was examined in the case of a non-Newtonian, power-law material (Carbopol solution), which contains Carbopol dust (0.13%), NaOH (0.05%), and water (99.82% by weight). The rheological law of the fluid can be described by $\tau = \mu_{PL}\dot{\gamma}^{n_{PL}}$,..
where the $\tau$ is the shear stress, $\dot{\gamma}$ is the shear rate, $\mu_{PL} = 0.1334P\alpha s^{nPL}$ is the dynamic viscosity, and $n_{PL} = 0.7266$ is the flow behaviour index. The mixer consists of two, tangential fluid jets, which drive the rotating motion of the fluid body. The distance between the top of the mixer and the jets is $0.184D$ ($D = 0.435$ m), and the ratio of the height and diameter of the tank is $1 \leq H/D \leq 2$.

The hyperbolic profile cannot be observed in the tangential flow field in the case of the examined power-law fluid in the range of $v_{jet} = 3.52-8.11$ m/s ($Re_{jet} = 264 - 608$). Furthermore, the rigid-body-like tangential flow field evolves in the $-0.35 < r/R < 0.35$ range, the coefficients of determination if the curve fit are $R^2 > 0.995$.

Publications for Theses #3 and #4: T2, T4, T5, T6, T9, T10.
Chapter 4

Experimental investigation of jet breakup of Bingham plastic slurries in air

4.1 Introduction

Even nowadays, a significant portion of electric energy is produced in coal-fired power plants, in which the by-product is coal ash (mostly fly ash and bottom ash), see e.g. [21]. The residuary ash is often mixed with water in hydrodynamic mixers and pumped to the deposition site safely and eco-friendly. This mixing process often includes a pre-mixing stage, where the slurry jets blunder into the freely falling dry ash and it is essential to design this device in a way that when the slurry jets reach the dry ash column, they are already broken up into droplets, to ensure mixing with the dry ash rather than 'cutting' it. Moreover, ensuring droplet formation improves the efficiency of the mixing and also reduces the hazard of the plug formation. This study focuses on developing a correlation that allows the estimation of jet breakup length for slurry jets in air.

There are several experimental and numerical studies on jet breakup in the scientific literature. For example [22], [23] and [24] present detailed descriptions of the theory of distortion and disintegration of liquid streams. Several studies present the relationship between Reynolds number and Ohnesorge number, e.g.: [23], [24], [25] and [26] based on the instability mechanism, four distinct regions can be assigned. The first region is the Rayleigh breakup regime where the size of the droplets are in the same order that the jet radius and the aerodynamic interaction with the air is not significant, see e.g.: [23]. This region was investigated first by [27] but e.g. [28] deals with this breakup numerically and [29] experimentally. The second region is the first wind-induced (FWI) where non-axisymmetric oscillations occur resulting still in droplet sizes comparable to the jet radius, see e.g. [23]. Here the interaction with the ambient air promotes the instability mechanism. In the second wind-induced (SWI) regime, the size of the droplets is smaller than the jet diameter, see e.g. [24]. The above-mentioned three regions were investigated
by \[26\] numerically. The last region is the atomization which is described by e.g. \[30\] and \[31\]. \[25, 32, 33\] deals with the shaping of the nozzles, e.g. circular, rectangle, square, triangle and elliptical jets are examined. \[34\] presents results in the complete range of above-mentioned regimes.

4.2 Experimental set-up

Figure 4.1 presents a sketch of the experimental set-up. A centrifugal pump conveys the fluid through the system from a tank via the pipe. The mass flow rate is set by the control valve and measured using a metering tank, a weighing-scale and a stop-watch. The fluid discharges downwards from a nozzle with 10 mm inner diameter toward the tank. During the fall, the jet breaks up into droplets. To identify the breakup length, 8 photos were shot for each parameter set. A length scale was also placed close to the jet to easily measure the breakup length.

![Figure 4.1: The sketch of the experimental rig.](image)

4.3 Material properties and dimensionless parameters

The ash was mixed with water and 9 solutions with different \(C_m = m_{ash}/m_{tot}\) concentrations (by mass) were used (\(m_{ash}\) is the mass of the ash and \(m_{tot}\) is the total mass of the slurry). The measurement of the rheology was performed on a Rheotest 2 RV2 rotational viscometer. The fluid displays Bingham plastic behaviour. After the evaluation of the experiments, the following equations can be written: \(\mu_{slurry} = 0.04014C_m + 0.01447\) (Pa s) and \(\tau_0 = 2.181C_m + 0.01447\) (Pa). The jet breakup length can be described with
the Weber-, Reynolds-, Froude- and Ohnesorge-numbers in the case of Newtonian fluids ([35, 36]). The fifth dimensionless parameter is the Hedström number describing the Bingham plastic behaviour.

4.4 Results

The ash-to-water concentration varies between appr. 9% and 56%. Five dimensionless formulae can have influence on the process from which the Froude- and the Weber-numbers dominate. With the help of the method of least squares, explicit equations are given to estimate the dimensionless jet breakup length, given in the Theses #5 and #6.

Thesis #5

Jet breakup and material properties of coal ash slurries were examined by experimental methods, for Bingham plastic fluids. The dimensionless parameter ranges are $He = 172 - 235$ for the Hedström number, $983 - 46866$ for the Weber number, $906 - 7169$ for the Reynolds number, and $7.06 - 42.13$ for the Froude number, respect to discharge. A wide concentration range (from 8.98% up to approx. 56% by mass) the dimensionless breakup lengths can be described by the $L/D = 1.847\sqrt{We} - 8.37Fr + 0.0271Re - 4104Oh - 1.375He + 471.9$, with the help of the method of least squares. The coefficient of determination of coefficients of the fit is $R^2 = 0.890$; $Oh = \sqrt{We}/Re$. The dimensionless breakup lengths can be described by the equation of $L/D = 0.1668\sqrt{We} + 2.979Fr + 62.66$ for engineering applications. The coefficient of determination of the fit is $R^2 = 0.887$ in this case.

Publication for Thesis #5: T8.
Own publications


[T8] Csizmadia P. és Hős Cs.: Nemnewtoni folyadék-levegő szabadsgugár felbomlási hosszának kíséreti vizsgálata, OGÉT 2014. XXII. Nemzetközi Gépészeti Találkozó, pp. 78-81

[T9] Csippa B. és Csizmadia P.: Hengeres tartályban erősen forgó áramlás numerikus
vizsgálata, OGÉT 2014. XXII. Nemzetközi Gépészeti Találkozó, pp.74-77

[T10] Csizmadia P. és Hős Cs.: Hengeres tartályban forgó áramlás időfüggő numerikus vizsgálata, OGÉT 2016. XXIV. Nemzetközi Gépészeti Találkozó, pp. 80-83


