CHAM Limited

Pioneering CFD Software for Education & Industry

CONCENTRATION DISTRIBUTION AND PRESSURE GRADIENT OF PARTICLE-WATER SLURRY FLOWS IN HORIZONTAL PIPES

by

Gianandrea Vittorio Messa, IIAR Dept., Polytechnic School of Milan, Milan, Italy

(Supervisor: Prof. Stefano Malavasi)

Introduction

During a second period of stay at CHAM Ltd. in the context of my PhD in Hydraulics as part of the research group of Prof. Stefano Malavasi, I went on working on the numerical modeling of slurry flow of water and solid particles horizontal pipes. Such flows are commonly encountered in many engineering applications, such as chemical, oil, mining and construction industries. Pressure gradient and concentration distribution have been the most serious concern of researchers, as they dictate the selection of pump capacity and may be used to determine parameters of direct importance (mixture and solid flow rates) as well as secondary effects like wall abrasion and particle degradation.

The high economic and technical burden of experimental tests and the limitations of the existing simplified models in global formulation made CFD an attractive alternative in recent years. However, the numerical models considered in previous studies either appear not very accurate under certain flow conditions [1] or require very long simulation time [2], which makes them unsuitable for the applications; therefore, the development of a suitable numerical model is still an open issue. This is basically the topic of my PhD Thesis and the object of my work during the periods at CHAM Ltd.

Starting from a literature search, I customized the IPSA model available in PHOENICS, adding specific constitutive equations and boundary conditions. The results – in terms of both concentration distribution and pressure gradient – were compared to experimental data available in literature [3,4,5], over a wide range of operating conditions: average solids concentration between 10% and 40% by volume; uniform particle size between 90 and 520 µm; slurry velocities between 1 m/s and 5.5 m/s; and pipe diameters between 50 and 150 mm.

Numerical model and boundary conditions

The Two-Fluid model was obtained by adding the following two features to the original IPSA model, necessary to correctly reproduce the flow:

- Mixture Viscosity: a correlation for the viscosity of the mixture, used to define the particle Reynolds number, is implemented. Among the different expressions available in literature, use is made of that of Mooney [6], which best fits the experimental data:

$$\mu_m = \rho_C \nu_{l,C} \exp\left(\frac{\eta\ \alpha_p}{1-\alpha_p / \alpha_{pm}}\right) \tag{1}$$

  in which η is the intrinsic viscosity, taken equal to 2.5 as suggested for spherical particles; α_{pm} is the maximum packing concentration, taken as 0.7; ρ_C is the density of the carrier fluid phase; and $\nu_{l,C}$ is the laminar kinematic viscosity of the carrier fluid phase.

- Drag force: the drag force law is related to the particle Reynolds number according to the Standard Drag Law correlation available in PHOENICS, but the particle Reynolds number is defined with respect to the viscosity of the mixture instead of that of the carrier fluid phase; therefore, $\mathrm{Re}_p = \rho_C d_p |\mathbf{U}_r| / \mu_m$, where d_p and $\mathbf{U}_r$ are the particle diameter and the slip velocity vector respectively. This modification is necessary to describe the phenomenon whenever in some cells the solid volume fraction approaches the maximum packing one.

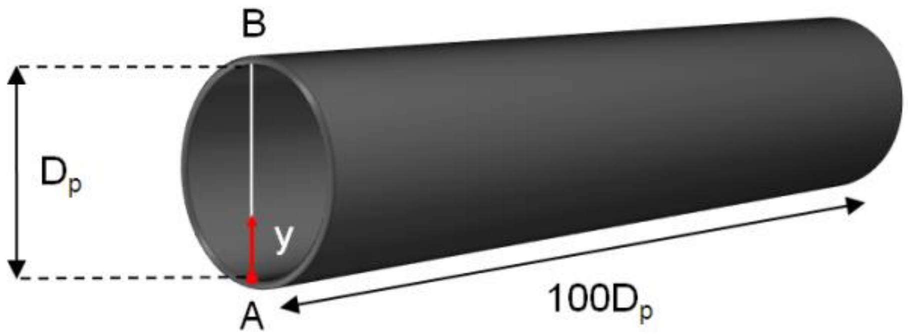

Figure 1 Sketch of the problem

Figure 1 shows the geometry of the problem; the computational domain covers only half of the pipe section due to a substantial symmetry of the phenomenon. A fully-developed turbulent flow profile is applied at the inlet. No slip is assumed between the phases. The inlet volume fraction of the solids is taken as uniform. At the outlet, the normal gradients of all variables and the value of the pressure are set to zero. The length of the computational domain, equal to 100 pipe diameters, is sufficient to ensure the reaching of fully-developed flow conditions.

At the pipe wall, no slip conditions are imposed to the carrier fluid phase, and a logarithmic law wall function is applied in the near wall-region. The proper wall boundary conditions for the solid phase are still a matter of discussion in literature. Two alternatives have been considered. At first, a zero-flux condition is applied to the particles. Afterwards, to account for particle-particle and particle-wall interactions, a Bagnold-type shear stress is imposed. In particular, the following term, derived from the model of Shook and Bartosik [7], is introduced in the momentum equation of the particle phase:

$$\tau_B = \left(\frac{8.3018}{\mathrm{Re}^{2.317}}\right)\rho_p d_p^2 \left[\left(\frac{\alpha_{pm}}{\alpha_p}\right)^{1/3} - 1\right]^{-1.5} \left(\frac{\tau_{w,C}}{\rho_C \nu_{l,C}}\right)^2 \qquad (2)$$

in which: $\mathrm{Re} = D_p U_s / \nu_{l,C}$ is the bulk Reynolds number, defined with respect to the pipe diameter D_p and the slurry bulk-mean velocity U_s; ρ_p is the density of the particles; and $\tau_{w,C}$ is the wall shear stress of the liquid phase.

Results

When imposing a zero-flux condition at the wall to the particle phase, the predictions of the Two-Fluid model show good agreement to the experimental evidence in terms of solid volume fraction distribution. As an example, Figure 2 reports the results for the flow conditions considered by Gillies et al. [4], i.e. $D_p = 0.1027$ m, $\rho_p = 2650$ kg/m^3, $d_p = 270$ μm, $U_s = 2.6$ m/s and mean solid volume fraction from 12% to 41%. The contour plots of Figure 2 highlight the gradual accumulation of the particles as the mean solid volume fraction increases, phenomenon that can be correctly reproduced applying the above mentioned modifications to the original IPSA model. The solid volume fraction profiles along the vertical diameter (AB in Figure 1) appear in quantitative agreement to the experimental data of Gillies et al. [4], as shown in Figure 3.

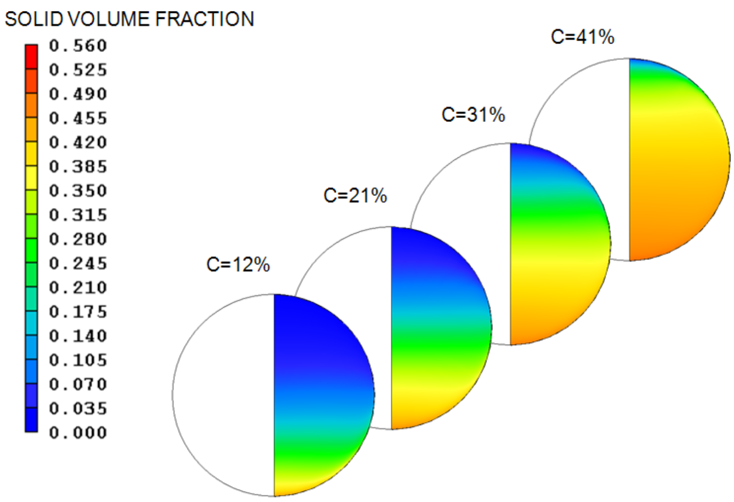

Figure 2 Contour plots for particle volume fraction

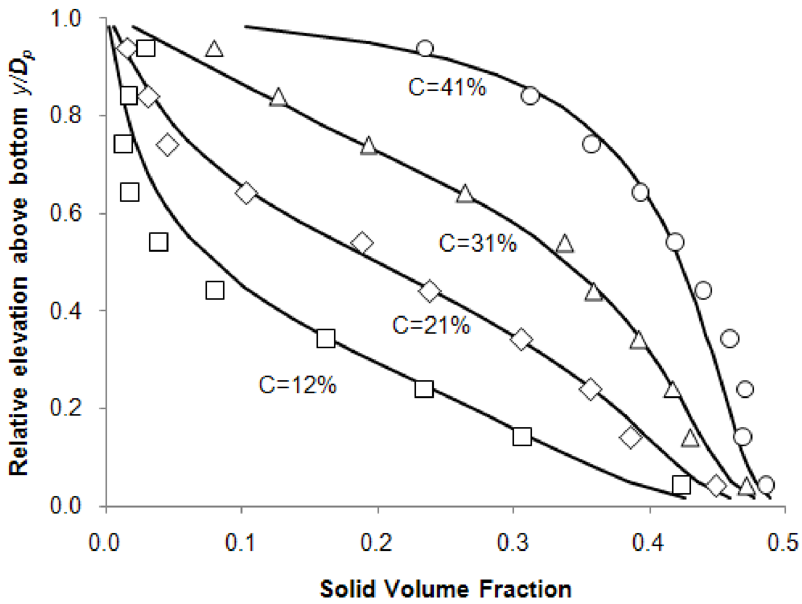

Figure 3 Particle concentration profiles: comparison between numerical predictions and experimental data of Gillies et al. [4].

The wall boundary condition of the solid phase is the key parameter affecting the friction losses. When imposing a zero-flux condition, the model is not capable to reproduce the increase in pressure losses due to the presence of the particles, resulting in an underestimation of the pressure gradient which increases with the particle concentration, reaching up to about 50% for high concentrated slurries.

The introduction of the Bagnold stresses term (Eq. 2) in the momentum equation of the solid phase allows catching the dependence of the pressure losses upon the particle concentration, keeping the underestimation of the pressure gradient below 30% for all the flow conditions considered. However, such improvement is often associated by a worsening of the concentration profile. Moreover, none of the two boundary conditions is capable to reproduce the increase in pressure losses occurring when the velocity is lower than the limit deposit velocity and a moving bed of particle forms at the bottom of the pipe. In such case, in fact, the phenomenon involves different physical mechanisms that the model does not account for.

Figure 4 reports the pressure gradient versus the slurry bulk-mean velocity for the case of $D_p = 0.1027$ m, $\rho_p = 2650$ kg/m^3, $d_p = 270$ μm and $C = 21\%$, comparing the experimental data of Gillies et al. [4] to the numerical predictions obtained using the two boundary conditions.

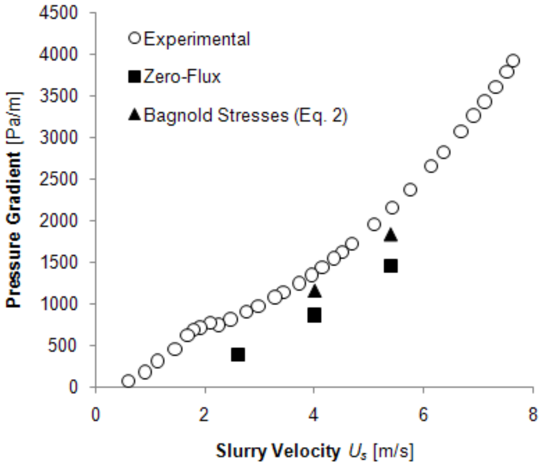

Figure 4. Pressure gradient versus slurry bulk-mean velocity: comparison between the experimental data of Gillies et al. [4] and the numerical predictions obtained using two boundary conditions

References

Lahiri, S.K., Gantha (2010). Slurry Flow Modelling by CFD. *Chem Ind. & Chem. Eng. Quaterly*, **16(4)**, 295-308.

Ekambara K., Sanders, R.S., Nandakumar, K., Masliyah, J.H. (2009). Hydrodynamic Simulation of Horizontal Slurry Pipeline Flow Using ANSYS-CFX. *Ind. Eng. Chem. Res.* **48(17)**, 8159-8171.

Roco, M.C., Shook, C.A. (1983). Modeling of Slurry Flow: The Effect of Particle Size. *Canadian J. Chem. Eng.* **61(4)**, 494-503.

Gillies, R.G., Shook, C.A., Xu, J. (2004). Modelling Heterogeneous Slurry Flow at High Velocities. *Canadian J. Chem. Eng.* **82(5)**, 1060-1065.

Kaushal, D.R., Tomita, Y. (2007). Experimental investigation for near-wall lift of coarser particles in slurry pipeline using γ-ray densitometer. *Powder Tech.* **172(3)**, 177-187.

Mooney, M.J. (1931). Explicit formulas for slip and fluidity. *J. Rheology* **2**, 210-222.

Shook, C.A., Bartosik, A.S. (1994). Particle-wall stresses in vertical slurry flows. *Powder Tech.* **81(2)**, 117-124.