Simulation of a Non-Newtonian Dense Granular Suspension in a Microfluidic Contraction

Andreas Mark[1], Gustaf Mårtensson[2,3], Thomas Kurian[3] and Fredrik Edelvik[1]

[1]*Fraunhofer-Chalmers Research Centre, Göteborg, Sweden*
[2]*BioNano Systems Laboratory, MC2, Chalmers University of Technology, Göteborg, Sweden*
[3]*Micronic Mydata AB, Täby, Sweden*

Keywords: granular suspension, contraction flow, computational fluid dynamics, rheology, shear thinning

Abstract

The success of a solder paste jet printer is based on an uninterrupted flow of fluid, specifically dense fluid suspensions, through a series of ducts inside the printing head. It is well known that the flow of dense suspensions is prone to jamming and sedimentation effects, both of which could entail detrimental failure modes in the printing heads. A thorough understanding of the fluid dynamics of suspensions as they flow through ducts and connections is of utmost importance. The purpose of this study is to propose a novel simulation framework and to show that it captures the main effects such as mass flow and partial jamming in a cylindrical duct test configuration. The granular suspension is a generic solder paste with solder particles immersed in a flux.

The simulations are performed in the multi-phase flow solver IBOFlow. A two fluid model is used for the granular suspension and the discretization is done an Euler-Euler framework. The averaged momentum equations from Enwald et al. (1996) are solved together with the common continuity equation generating a shared pressure field. Explicit constitutive equations for the interfacial momentum transfer and particle pressure are employed. To capture the shear thinning effects of the non-Newtonian suspensions the standard Carreau rheology model is used.

To study how the fluid flow affects the local volume fraction and partial jamming in the duct, simulations are performed for different applied pressure drops ranging from one to five bars. For both particle pressure models, the resulting mean bulk velocities are compared with experiments with good agreement, and partial jamming is observed. Hence, it is concluded that the proposed framework is suitable to model and simulate the granular suspension in a micro fluid contraction.

Introduction

The flow of dense suspensions through gradual or sudden expansions and contractions is of considerable industrial and academic interest. Examples of industrial applications include separation apparatus, paper dewatering, jetting heads and pharmaceutical systems. Areas of interest in these applications are for example the effect of geometrical configurations on pressure drop, localized aggregation of particles over the system and intermittent flow speed and stress fluctuations. A thorough understanding of the fluid dynamics of suspensions as they flow through ducts and connections is of utmost importance. A majority of the research concerning contraction and expansion is focused on single-phase Newtonian and non-Newtonian flows, see for example Astarita and Greco (1968) and Boger (1987).

For the specific case of dense suspensions, only limited work is found in the literature. Yaras et al. (1994) studied the dynamics of pressure-driven flow through a capillary with very dense suspensions. They found that filtration effects in the flow resulted in concentration gradients in the suspension that were correlated with time-periodic pressure fluctuations. Moraczewski and Shapley (2007) utilized nuclear magnetic resonance (NMR) imaging to study the flow of dense suspensions through an abrupt axisymmetric contraction-expansion. The pressure drop over the test section was found to be lower than would be expected with a uniform Newtonian fluid. This behaviour was attributed to shear-induced particle migration.

Kulkarni et al. (2010) combined an experimental study of gravity driven flow of dense suspensions through an abrupt area contraction, and developed an approximate model for this case. The change in pressure over the area contraction is related to the particle pressure and a model tied to self-filtration over the contraction is proposed. A number of other studies have shown that the particle pressure is the dominant mechanism underlying particle migration, see for example Sierou and Brady (2002) and Yurkovetsky and Morris (2008).

The purpose of this study is to develop a novel simulation framework and to apply it to the flow of dense suspensions in a cylindrical contraction. Furthermore, to validate with experiments that the proposed framework captures the main effects, such as mass flow and partial jamming.

Experimental setup

The geometrical configuration for the experiment in this study consists of a cylindrical duct with a diameter of d_1 =

0.8 mm that contracts via a right circular cone with an apex angle of $\alpha = 60^\circ$ to a circular duct with a diameter of d_2 = 0.2 mm, see Figure 1. The contraction was manufactured in Ultem polyetherimide resin in order to ensure a fine surface finish, chemical resistance and visual accessibility.

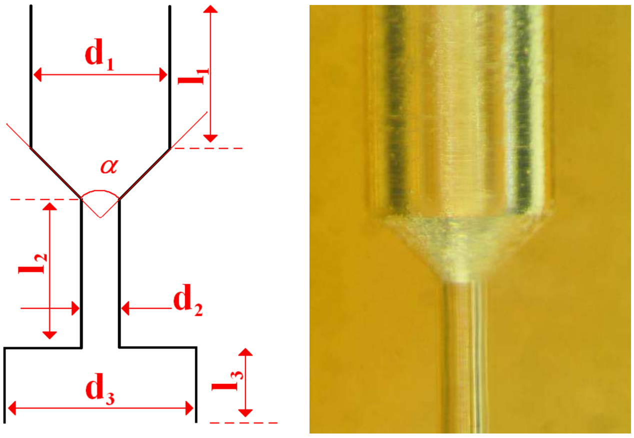

Figure 1 : Layout and image of the experimental micro channel contraction.

A suspension consisting of an organic resin-based carrier fluid and SnAgCu-alloy spheres was used for the contraction experiments. The spherical particles range in size between 10 and 25 μm, see Figure 2.

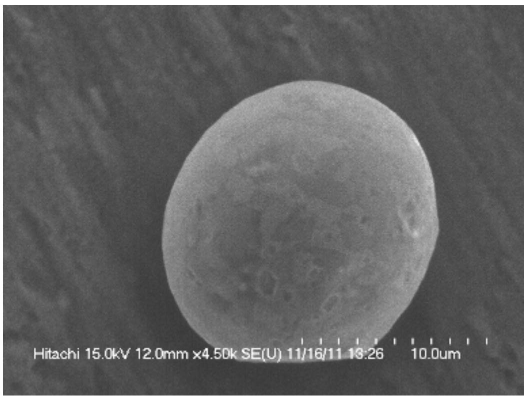

Figure 2 : An example of a granule contained in the suspension.

The fluid is shear-thinning, see Figure 3, and the viscosity decreases by three orders of magnitude for a four orders of magnitude increase of shear rate.

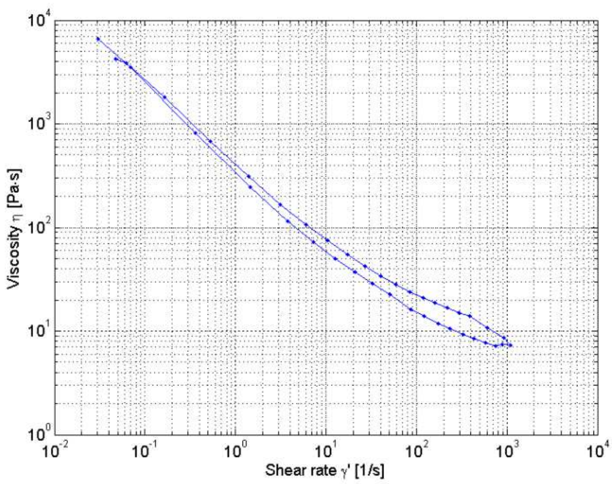

Figure 3 : Viscosity as a function of shear rate for the suspension used in the experiment.

A pressure gradient was introduced over the geometric configuration and the mass flow of the suspension passing through the duct was measured by collecting the fluid during a time period of 300 s. The flow through the duct was also filmed using a digital camera in order to document time-dependent changes in the flow.

Models and Numerical Schemes

The simulations are performed with IBOFlow (2011), a multi-phase flow solver developed at the Fraunhofer-Chalmers Centre. A two-fluid model is used for the granular suspension and the discretization is done in an Euler-Euler framework. The averaged momentum equations from Enwald et al. (1996) is solved together with the common continuity equation generating a shared pressure field. Explicit constitutive equations for the interfacial momentum transfer model from Gidaspow (1994) and the particle pressure from Zenit et al. (1997) and Kulkarni et al. (2010) are employed. To capture the shear thinning effects of the non-Newtonian suspensions, the rheology is modeled by a Carreau model.

Granular models

The granular-suspension is modelled by a two fluid model discretized in an Euler-Euler framework. The continuity equations of the two phases are

$$\frac{\partial \alpha_f}{\partial t} + \nabla \cdot \left(\alpha_f \vec{v}_f\right) = 0 \tag{1}$$

$$\frac{\partial \alpha_s}{\partial t} + \nabla \cdot \left(\alpha_s \vec{v}_s\right) = 0 \tag{2}$$

where $()_f$ denotes the fluid phase, $()_s$ denotes the solid or granular phase, α is the volume fraction and $\vec{v}$ represents the averaged velocity field. The averaged momentum equations from Enwald et al. (1999) are employed,

$$\rho_f \alpha_f \left[\frac{\partial \vec{v}_f}{\partial t} + \vec{v}_f \cdot \nabla \vec{v}_f\right] = -\alpha_f \nabla P + \nabla \cdot \alpha_f \tau_f + \alpha_f \rho_f \vec{g} - \beta\left(\vec{v}_f - \vec{v}_s\right) \tag{3}$$

$$\rho_s \alpha_s \left[\frac{\partial \vec{v}_s}{\partial t} + \vec{v}_s \cdot \nabla \vec{v}_s\right] = -\alpha_s \nabla P + \nabla \cdot \alpha_s \tau_s - \nabla P_s + \alpha_s \rho_s \vec{g} + \beta\left(\vec{v}_f - \vec{v}_s\right) \tag{4}$$

where ρ is the density, P is the common pressure, τ is the stress tensor, $\vec{g}$ is the gravity, β is the interphase transfer coefficient and P_s the particle pressure. To derive the common pressure equation, the divergence of the total mean velocity field is set to zero

$$\nabla \cdot \vec{v} = \nabla\left(\alpha_f \vec{v}_f + \alpha_s \vec{v}_s\right) = 0 \tag{5}$$

and the momentum equations are inserted in a SIMPLEC manner. Hence, the momentum equations and the pressure equations are solved segregated. The volume fractions are convected in the following conservative form proposed by Weller (2002),

$$\frac{\partial \alpha_f}{\partial t} + \nabla \cdot \left(\alpha_f \vec{v}_f\right) + \nabla \cdot \left(\vec{v}_f - \vec{v}_s\right)\alpha_f \alpha_s = 0 \tag{6}$$

where the chock capturing convective scheme CICSAM developed by Ubbink (1997) is employed.

Constitutive models

To close the granular model constitutive models are required. For the particle pressure two different models are tested. One proposed by Fuscolo and Gibilaro (1987),

$$P_s = \alpha_s^2 \rho_s u_e^2 \tag{7}$$

where u_e the elastic wave velocity is defined as

$$u_e = \left(3.2 g d_s \frac{\rho_s - \rho_f}{\rho_s} \right)^{1/2} \tag{8}$$

where d_s is the granular diameter. This model was initially developed for dense fluidized flows, but seems to work well for the micro contraction. The other model considered is the one proposed by Kulkarni and co-workers (2010). For a non-Brownian suspension the particle pressure is expressed as

$$P_s = \mu_n \dot{\gamma} \tag{9}$$

where $\dot{\gamma}$ is the local shear rate. The normal stress viscosity is modeled as

$$\mu_n \sim 0.75 \left(\frac{\alpha_s}{\alpha_s^{\max}} \right)^2 \left(1 - \frac{\alpha_s}{\alpha_s^{\max}} \right)^{-2} \tag{10}$$

where $\alpha_s^{\max}$ is the maximum granular volume fraction.

The interphase transfer coefficient from Gidaspow (1994) is employed

$$\beta = \begin{cases} 150 \dfrac{\alpha_s^2 \mu_f}{\alpha_f d_s^2} + \dfrac{7}{4} \dfrac{\alpha_s \left| \vec{v}_f - \vec{v}_s \right|}{d_s} & \text{if } \alpha_s > 0.2 \\ \dfrac{3}{4} C_d \dfrac{\alpha_f \alpha_s \rho_f \left| \vec{v}_f - \vec{v}_s \right|}{d_s} \alpha_f^{-2.65} & \text{if } \alpha_s \le 0.2 \end{cases} \tag{11}$$

where μ_f is the fluid viscosity. The empirical drag coefficient is defined as

$$C_d = \begin{cases} \dfrac{24}{\mathrm{Re}_p \alpha_f} \left[1 + 0.15 \left(\alpha_f \mathrm{Re}_p \right)^{0.687} \right] & \text{if } \alpha_f \mathrm{Re}_p < 1000 \\ 0.44 & \text{if } \alpha_f \mathrm{Re}_p \ge 1000 \end{cases} \tag{12}$$

where the Reynold's particle number is given by

$$\mathrm{Re}_p = \frac{d_s \rho_f \left| \vec{v}_f - \vec{v}_s \right|}{\mu_f}. \tag{13}$$

In the present work the Carreu rheology model from Bird et al. (1987) is used. The viscosity is therefore determined by

$$\mu = \mu_0 \left(1.0 + \left(\lambda \dot{\gamma} \right)^2 \right)^{0.5(N-1)} \tag{14}$$

where μ_0 is the zero shear viscosity, λ is the time constant, and N is the power law exponent. For different materials these constants are determined by experiments.

Flow solver

The simulations are performed with IBOFlow (Immersed Boundary Octree Flow Solver), the multi-phase flow solver developed at the Fraunhofer-Chalmers Centre. In IBOFlow, all equations are discretized by the finite volume method on a Cartesian octree grid that can be dynamically refined and coarsened. All variables are stored in a co-located arrangement and the momentum equations and the pressure equation is solved in a segregated manner. The Backward Euler scheme is used for the temporal discretization. Further, the hybrid immersed boundary method developed by Mark and co-workers (2008, 2011) is used to model the presence of solid objects, without the need of a body-fitted mesh. In the method the fluid velocity is set to the local velocity of the object with an immersed boundary condition. To set this boundary condition a cell type is assigned to each cell in the fluid domain. The cells are marked as fluid cells, extrapolation cells, internal cells or mirroring cells depending on the position relative to the IB. The velocity in the internal cells is set to the velocity of the immersed object with a Dirichlet boundary condition. The extrapolation and mirroring cells are used to construct implicit boundary conditions that are added to the operator for the momentum equations. This results in a fictitious fluid velocity field inside the immersed object. Mass conservation is ensured by excluding the fictitious velocity field in the discretized continuity equation. The result is a robust method that is second order accurate in space and implicitly formulated.

Results and Discussion

Simulations on the micro duct are performed using the presented constitutive models. For the rheology the Carreau model is employed both for the granular and the carrying fluid phase. In Table 1, the simulation parameters are presented.

Table 1 : Simulation parameters.

Variable name	Symbol	Value	Unit
Time step	Δt	0.1, 0.025	ms
Fluid density	ρ_f	1000	kg/m^3
Solid density	ρ_s	3000	kg/m^3
Granular diameter	d_s	15	μm
Applied pressure	-	0.5-5.0	Bar
Time constant	λ	-1.43 10^8	S
Zero shear viscosity	μ_0	5.04 10^8	$Pa\ s$
Power law exponent	N	0.276	-
Max. solid volume fraction	$\alpha_s^{\max}$	0.65	-

To validate the framework the flow through the contraction is simulated for different applied pressure drops and for the two particle pressure models. Two different time steps were employed; the smaller one was required for the Kulkarni pressure model. At the top inlet, the pressure drop is applied and the volume fraction is set to 0.5. The mean granular velocity is averaged between 0.02 and 0.1s. The fluid grid is refined two times at the edge of the cylindrical duct, generating a cell size between 12.5 and 50.0 μm, see Figure 4.

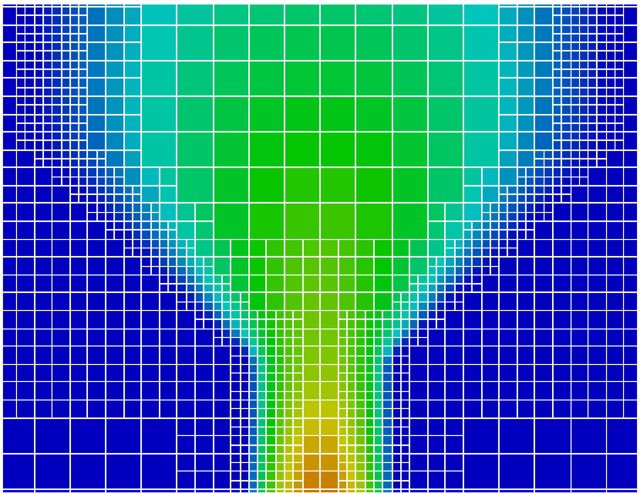

Figure 4 : The refined fluid octree grid along with the fluid phase velocity field are shown.

In Figure 5 a snapshot of a simulation with an applied pressure drop of $\Delta p = 2$ bar is shown. In the left of the Figure the volume fraction shows how the granular phase is compressed at the contraction. This compression could eventually lead to a partial or full jamming. At the contraction a maximum granular volume fraction between 0.57 and 0.59 is observed. In the next part of the Figure the pressure is visualized. Here it is clearly seen that the pressure drop is concentrated over the pipe with a smaller radius, due to the fact that the fluid viscous forces are dominating there. In the middle, the granular velocity field is shown. In the transient simulation it is noticed that initially the velocity is higher and then it decreases with an oscillating movement which comes from the rheology model and the interphase momentum transfer. In the next duct the velocity of the carrying fluid is shown, which also oscillates as the viscosity rapidly changes in the rheology model. In the rightmost duct the viscosity of the carrying fluid is shown. Notice that a log scale has been adopted to capture the large differences.

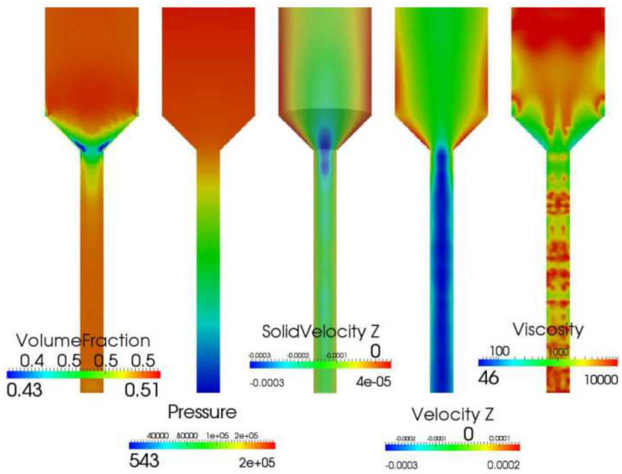

Figure 5 : Simulation of the granular suspension with an applied pressure drop $\Delta p = 2$ bar with Foscolo's pressure model.

In Figure 6, the simulated and experimental mean granular or bulk velocity are plotted against the applied pressure drop. In the figure it is noticed that the simulations and experiments lie remarkably close to each other. For low pressure drops the two particle pressure models generate almost the same solution. As the pressure drop increases and the volume fraction compression at the contraction increases the particle pressure models become more important. The differences between the models are clearly seen for higher pressure drops. Further, the Kulkarni pressure model is more aggressive and operates in a smaller region in space and therefore a smaller time step is required.

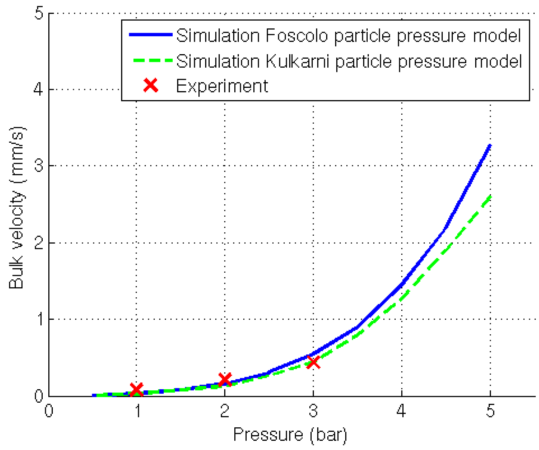

Figure 6 : The simulated and experimental bulk or the mean velocities of the granular particles are plotted against the applied pressure.

Conclusions

In this study, a model to simulate the granular suspension is proposed. Through simulation it is also shown that the model captures the main behaviour of the granular suspension. For two different pressure models the granular velocities are compared to performed experiments with good agreement. Further partial and total jamming have been observed during test simulations. Hence, it is concluded that the proposed framework is suitable to model and simulate the granular suspension in a microfluidic contraction.

Acknowledgements

The authors would like to thank Dr. Tord Karlin for insightful conversations during the project. This material is based on work supported by the Swedish Research Council under Grant No. 621-010-4334.

References

Astarita, G. and Greco, G. 1968 "Excess pressure drop in laminar flow through sudden contraction—Newtonian liquids," *Ind. Eng. Chem. Fundam.*, **7**, 27.

Bird, R.B. and Armstrong, R.C. and Hassager, 0. "Dynamics of polymer liquids" vol. 1, 2nd ed. Wiley, New York., 1987.

Boger, D.V. 1987 "Viscoelastic flows through contractions," *Annu. Rev. Fluid Mech.*, **19**, 157.

Boyer, F., Guazzelli, E. and Pouliquen, O. 2011 "Unifying Suspension and Granular Rheology", *PRL*, **103**, 188301.

Van Doormaal, J.P. and Raithby, G.D. 1984 "Enhancements of the SIMPLE method for predicting incompressible fluid flows", *Num. Heat Transfer*, **7**:147–163.

Enwald, H., Peirano, E. and Almstedt, A.E. 1996 ''Eulerian Two-Phase Flow Theory Applied to Fluidization,'' *Int. J. Multiphase Flow*, **22**, 21.

Fuscolo, P.U. And Gibilaro, L.G. 1987 "Fluid Dynamic Stability of Fluidised Suspensions: The Particle Bed Model", *Chem. Eng. Sc.*,**42**:1489.

Gidaspow, D., "Multiphase Flow and Fluidization", Academic Press, San Diego 1994.

Kulkarni, S.D., Metzger, B. and Morris, J.F. 2010 "Particle-pressure-induced self-filtration in concentrated suspensions". *Phys. Rev.*, **82**:010402.

Mark, A and Rundqvist, R and Edelvik, F. 2011 "Comparison Between Different Immersed Boundary Conditions for Simulation of Complex Fluid Flows", *FDMP*, **7**:3:241.

Mark, A., van Wachem, B. G. M., 2008 *Journal of Computational Physics* 227:6660-6680.

Parry, A.J. and Millet, O. 2010 "Modeling Blockage of Particles in Conduit Constrictions: Dense Granular-Suspension Flow", *ASME J. Fluids Eng.*, **132**, 011302.

Ubbink. O. "Numerical prediction of two fluid systems with sharp interfaces", PhD.Thesis, Department of Mechanical Engineering, Imperial College of Science, London, United Kingdom, 1997.

Weller, H.G. "Derivation, modelling and solution for the conditionally averaged two-phase flow Equations". Technical Report, TR/HGW/02:Nabla Ltd., 2002.

Yaras, P., Kalyon, D.M. and Yilmazer, U. 1994 "Flow instabilities in capillary flow of concentrated suspensions", *Rheol. Acta*, **33**, pp. 48-59.

Zenit, R. Hunt, M. L. and Brenner, C. E. 1997 "Collisonal particle pressure measurements in solid-liquid flows", *J. Fluid. Mech.* **353**, 261.