Proceedings of the 12ᵗʰ International Conference on
Computational Fluid Dynamics in the Oil & Gas,
Metallurgical and Process Industries

Progress in Applied CFD – CFD2017
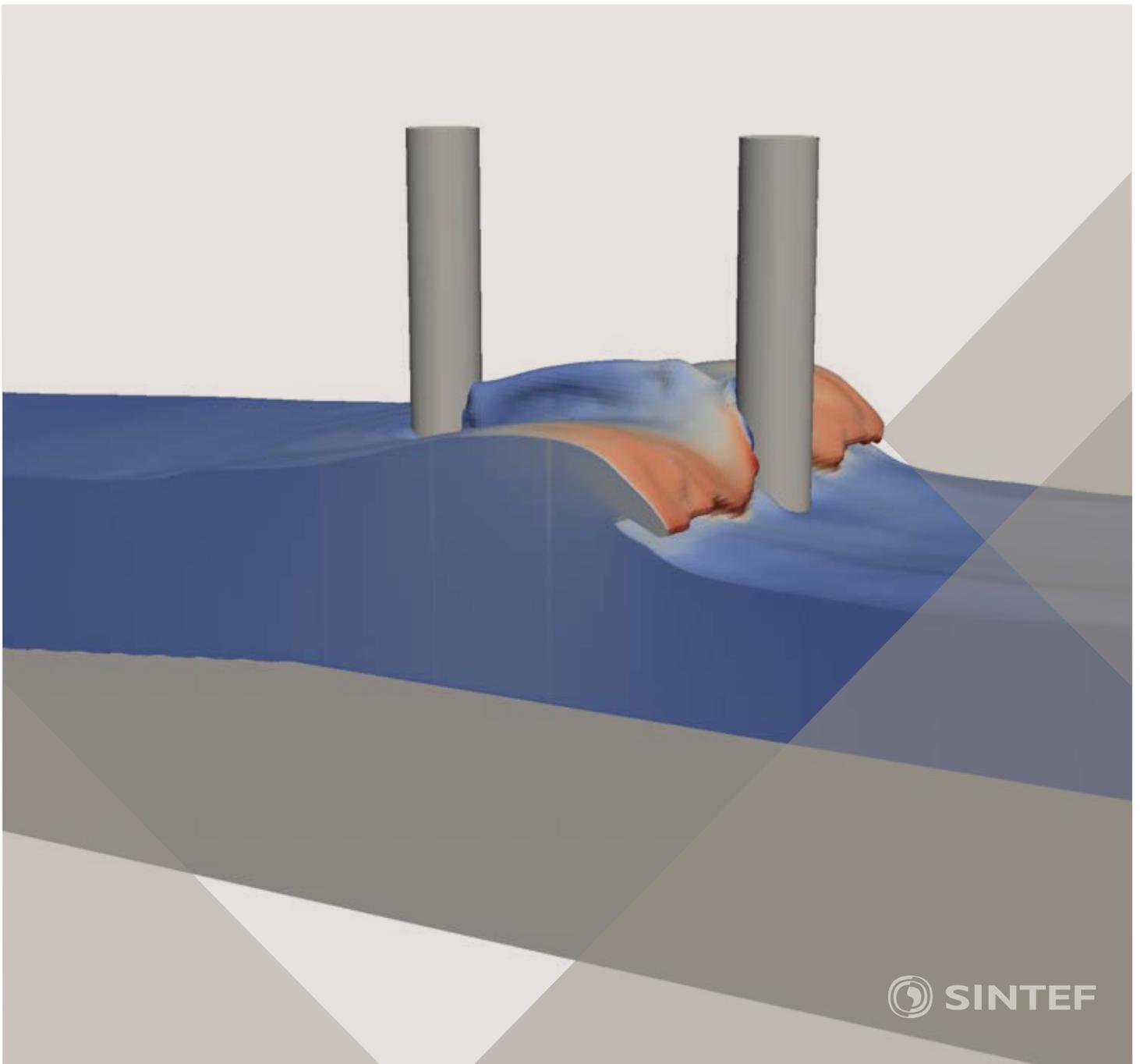
SINTEF

SINTEF Proceedings

Editors:
Jan Erik Olsen and Stein Tore Johansen

Progress in Applied CFD – CFD2017

Proceedings of the 12th International Conference on Computational Fluid Dynamics
in the Oil & Gas, Metallurgical and Process Industries

SINTEF Academic Press

SINTEF Proceedings

SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings
on a variety of scientific topics.

The processes of peer-reviewing of papers published in SINTEF Proceedings are
administered by the conference organizers and proceedings editors. Detailed
procedures will vary according to custom and practice in each scientific community.

PREFACE

This book contains all manuscripts approved by the reviewers and the organizing committee of the 12th International Conference on Computational Fluid Dynamics in the Oil & Gas, Metallurgical and Process Industries. The conference was hosted by SINTEF in Trondheim in May/June 2017 and is also known as CFD2017 for short. The conference series was initiated by CSIRO and Phil Schwarz in 1997. So far the conference has been alternating between CSIRO in Melbourne and SINTEF in Trondheim. The conferences focuses on the application of CFD in the oil and gas industries, metal production, mineral processing, power generation, chemicals and other process industries. In addition pragmatic modelling concepts and bio-mechanical applications have become an important part of the conference. The papers in this book demonstrate the current progress in applied CFD.

The conference papers undergo a review process involving two experts. Only papers accepted by the reviewers are included in the proceedings. 108 contributions were presented at the conference together with six keynote presentations. A majority of these contributions are presented by their manuscript in this collection (a few were granted to present without an accompanying manuscript).

The organizing committee would like to thank everyone who has helped with review of manuscripts, all those who helped to promote the conference and all authors who have submitted scientific contributions. We are also grateful for the support from the conference sponsors: ANSYS, SFI Metal Production and NanoSim.

Stein Tore Johansen & Jan Erik Olsen

CONTENTS

12th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, NORWAY
May 30th - June 1st, 2017

CFD 2017

DIRECT NUMERICAL SIMULATION OF PROPPANT TRANSPORT IN A NARROW CHANNEL FOR HYDRAULIC FRACTURING APPLICATION

R. V. MAITRI[1]*, **I. KOIMTZOGLOU**[1], **S. DAS**[1], **J.A.M. KUIPERS**[1], **J.T. PADDING**[2], **E.A.J.F. PETERS**[1]

[1]Multiphase Reactors Group, Department of Chemical Engineering and Chemistry,

Eindhoven University of Technology, THE NETHERLANDS

[2]Intensified Reaction & Separation Systems, Process and Energy Department,

Delft University of Technology, THE NETHERLANDS

* E-mail: r.maitri@tue.nl

ABSTRACT

An efficient and accurate model for the direct numerical simulations (DNS) of liquid-solid flows is presented in this work. In this numerical model, fluid-solid coupling is achieved by implementing the no-slip boundary condition at the particles' surfaces by using a second order ghost-cell immersed boundary method, allowing for a fixed Cartesian grid to be used for solving the fluid equations. The particle-particle and particle-wall interactions are implemented using the soft sphere collision model. Lubrication forces are included through a sub-grid scale model because of its range of influence on a scale smaller than the grid size.

After the validation of the model, the transport of solid particles in a narrow channel is simulated to mimic the proppant transport in rock fractures in fracking process. The simulations are performed for solids volume fractions ranging from 1.7 to 20 % with the range of Reynolds and Archimedes number: 100-400 and 0-7848, respectively.

Keywords: Direct Numerical Simulation (DNS), Immersed Boundary Method (IBM), Multiphase flow, fracking

.

NOMENCLATURE

Greek Symbols

ε_s Solids volume fraction
μ Dynamic viscosity, $[kg/m.s]$
ξ_s Dimensionless distance
ρ Density, $[kg/m^3]$
τ Viscous stress, $[N/m^2]$
ϕ Variable in the equation to be solved
$\overline{\omega},\overline{\Omega}$ Rotational velocity, $[1/s]$

Latin Symbols

a Coefficients in discretized equation
Ar Archimedes number
b_c Explicit part in the discretized equation
D Diameter, $[m]$
$\overline{F}_{f \to s}$ Force exerted by fluid on solid, $[N]$
$\overline{F}_{s \to s}$ Force in solid-solid interaction, $[N]$
$\overline{g}$ Gravitational acceleration, $[m/s^2]$
H Height of the channel, $[m]$
I Moment of inertia, $[kg.m^2]$
m Mass, $[kg]$
$\overline{n}$ Unit normal vector,
N Total number of particles

p Pressure, $[N/m^2]$
$\overline{r}$ Position vector, $[m]$
Re Reynolds number
St Stokes number
t Time, $[s]$
$\overline{T}_{f \to s}$ Torque exerted by fluid on solid, $[N.m]$
$\overline{u}$ Fluid velocity, $[m/s]$
$\overline{w}$ Translational velocity, $[m/s]$
$\overline{y}$ Vertical height $[m]$

Sub/superscripts

f Fluid phase
p Particle
s Solid phase

Operators

∇ Gradient $[1/m]$
$\nabla \cdot$ Divergence $[1/m]$
∇^2 Laplace $[1/m^2]$

INTRODUCTION

Particle laden flows are encountered in many industrial as well as natural processes. These include proppant transport in fracking, biological flows, sediment transport in river and environmental flows. The fundamental understanding of fluid-solid multiphase flows is important for the optimization of these processes and computational fluid dynamics (CFD) is an effective numerical tool to obtain an insight in such complex processes.

For fluid-solid interaction, the immersed boundary method (Peskin, 1972) was introduced to couple the movement of the flexible membrane and the fluid around it. This method used the feedback forcing method to enforce the no-slip boundary condition on the particle surface. A different approach was proposed by (Fadlun *et al.*, 2000) to have a direct forcing to impose the no-slip boundary condition. (Uhlmann, 2005) combined the direct forcing method with the regularized delta function to remove oscillations in moving particles' simulation. This method was improved later (Breugem, 2012; Kempe and Fröhlich, 2012) to account for the lower solid to fluid density ratios and to improve the order of accuracy of the original method. Another efficient variant of the IBM named ghost cell method (Tseng and Ferziger, 2003) is also often used. Here the ghost node inside the solid is given a velocity to impose the no-slip boundary condition

on the particle surface. This method was later modified and extended for moving particles in fluidized beds (Deen *et al.*, 2012) and is used with further modification in this paper for the simulations.

Particle laden flows can be categorized in two classes: free-surface flow like sediment transport in a river and narrow channel flow like proppant transport in a rock fracture. The relevant length scales and the flow structures in both the phenomena are quite different to each other. This work focuses on the narrow channel flow to obtain an insight into the proppant transport phenomenon. Previous numerical studies of particle transport in a narrow channel were performed for 2D circular particles using Arbitrary-Lagrangian-Eulerian (ALE) method (Choi and Joseph, 2001; Patankar *et al.*, 2001). In this paper, fully resolved 3D simulations are performed to capture the effect of flow structures in the transverse direction as well.

MODEL DESCRIPTION

Our DNS model solves the coupled fluid-solid flow where the fluid phase is governed by continuity and Navier-Stokes equation and the solid motion is governed by Newton-Euler equations. The mathematical formulation of these equations is as follows (Eq. 1 - 4) :

Fluid phase:

The governing equations for incompressible Newtonian fluid flow are:

$$(\nabla \cdot \overline{u}) = 0 \tag{1}$$

$$\frac{\partial \rho_f \overline{u}}{dt} + (\nabla \cdot \rho_f \overline{uu}) = -\nabla p + \mu_f \nabla^2 \overline{u} + \rho_f \overline{g} \tag{2}$$

The viscous term in the Navier-Stokes equation is discretized with the standard second-order central difference scheme. For the convective terms, the total variation diminishing min-mod scheme is used, with a deferred correction. In the deferred correction, first order upwind (FOU) is implemented implicitly and the corrector step is carried out explicitly. The velocity and pressure variable are solved on a staggered grid with the standard fractional step method.

Solid phase:

The translational and rotational motion of particles is governed by the following equations:

$$m_p \frac{d\overline{w}_p}{dt} = m_p \overline{g} + \overline{F}_{f \to s} + \overline{F}_{s \to s} \tag{3}$$

$$I_p \frac{d\overline{\omega}_p}{dt} = \overline{T}_{f \to s} \tag{4}$$

The force and torque exerted by the fluid on a spherical particle is:

$$\overline{F}_{f \to s} = -\iint_{S_p} (\tau_f \cdot \overline{n} + p\overline{n}) dS \tag{5}$$

$$\overline{T}_{f \to s} = -\iint_{S_p} (\overline{r} - \overline{r}_p) \times (\tau_f \cdot \overline{n}) dS \tag{6}$$

The particle-particle interaction ($\overline{F}_{s \to s}$) is accounted for by the standard soft-sphere collision (Cundall and Strack, 1979) for the normal forces whereas a sub-grid scale lubrication model (Brenner, 1961) is used for the correction of the unresolved hydrodynamic interaction between particles.

Fluid-solid interaction

The fluid-solid interaction takes place through momentum exchange at the particle surface and is incorporated in this study using the second order ghost-cell immersed boundary method (Deen *et al.*, 2012; Das *et al.*, 2016). In this method, the no-slip boundary condition on the particle surface is enforced implicitly by modifying the coefficient matrix of the fluid velocities at the level of the discrete equivalent of Eq. 2 . In Fig. 1, ϕ_i denotes a flow variable in cell at position i, such as a velocity component. The velocity at ghost cell 0 in Fig. 1 is extrapolated based on the particle velocity and the neighbouring fluid velocities and is represented as:

$$\phi_0 = -\frac{2\xi_s}{1 - \xi_s}\phi_1 + \frac{\xi_s}{2 - \xi_s}\phi_2 + \frac{2}{(1 - \xi_s)(2 - \xi_s)}\phi_p \tag{7}$$

It can be noted that the velocity at point ϕ_1 and ϕ_2 are to be solved for the new time step as well and hence, the extrapolation modifies the coefficients for the equation of velocity at ϕ_1. The generic form of the discretized momentum equation and the updated coefficients are given in Eq. 8 - 11, where a_i and b_i indicate the coefficients and the explicit terms, respectively, before incorporating boundary conditions at the fluid-solid interface and $\hat{a}_i$ and $\hat{b}_i$ are the modified values after IBM implementation.

$$a_1 \phi_1 + \sum_{nb} a_{nb} \phi_{nb} = b_c \tag{8}$$

$$\hat{a}_1 = a_1 - a_0 \frac{2\xi_s}{1 - \xi_s} \tag{9}$$

$$\hat{a}_2 = a_2 + a_0 \frac{\xi_s}{2 - \xi_s} \tag{10}$$

$$\hat{b}_c = b_c - a_0 \frac{2}{(1 - \xi_s)(2 - \xi_s)}\phi_p \tag{11}$$
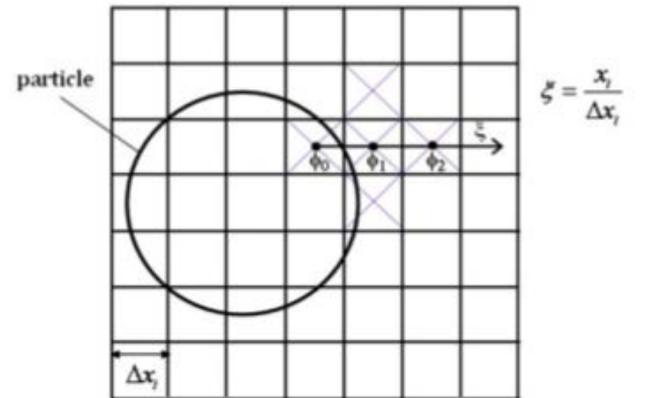
Figure 1: Representation of the fluid-solid interface on the Cartesian grid (Deen *et al.*, 2012)

VERIFICATION AND VALIDATION

The original IBM (Deen *et al.*, 2012) is modified to compute the torque on spherical particles accurately. The performance of the new algorithm is quantified by comparing the torque computations with the analytical solution of the slowly rotating sphere in a large body of quiescent fluid in a creeping flow regime (Fig. 2) and the sedimentation of a single particle in a box filled with the liquid.

Torque on a sphere

The torque on a sphere in a creeping flow (Bird *et al.*, 2007):

$$T = \pi \mu_f \omega_p D_p^3 \qquad (12)$$

The results of the improved IBM method are presented in the Table 1. It is found that the error associated with the torque computation is reduced by 3-13 times (depending on the resolution) after the implementation of an improved algorithm. The accurate torque computation is very important in the particle laden flows as the particle rotation affects the fluid flow which in turn affects the particles again and hence, the small error can amplify over the time.
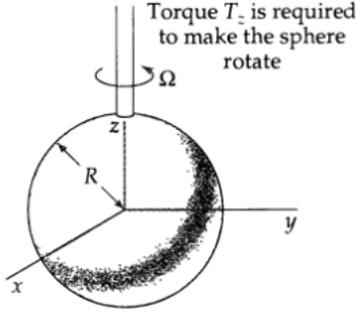
Figure 2: Freely rotating sphere in the quiescent fluid (Bird *et al.*, 2007)

Table 1: Error estimation of the numerical values of torque with the original and improved IBM ($\mu_f = 2$ kg/ m.s, $\omega = 10^{-5}\ s^{-1}$, $D_p = 0.2$ m), Torque$_{analytical} = 5.03 \times 10^{-7}$ N-m

$D_p/\Delta x$	Torque (N-m)		% Error	
	Original	Improved	Original	Improved
10	4.52×10^{-7}	4.86×10^{-7}	10.14 %	3.31 %
20	4.60×10^{-7}	4.96×10^{-7}	8.55 %	1.29 %
40	4.88×10^{-7}	5.00×10^{-7}	2.98 %	0.47 %
80	4.92×10^{-7}	5.02×10^{-7}	2.2 %	0.17 %

Sedimentation of a single particle

As a next validation case, numerical simulation of a single particle sedimentation in a large box is performed and the results are compared with the experimental measurements (Ten Cate *et al.*, 2002). The simulation is carried out in the domain of the size $6.67D_p \times 10.67D_p \times 6.67D_p$ with initial particle centre position at $8.5D_p$ from the bottom wall. The diameter of the particle is 0.015 m and the density is 1120 kg/m^3. Free-slip boundary condition is applied on the walls for velocity. The comparison results for the position of the bottom surface of the particle from bottom wall and its vertical velocity are presented in Fig. 3. An excellent agreement between the simulation and experimental results is obtained.

RESULTS

To mimic the proppant transport phenomenon in rock fractures, multi-particle simulations in the narrow channel are performed in this paper. Especially, the influence of the initial particle configuration and the solids volume fraction is studied. The transport of particles is governed by the two important non-dimensional numbers:

Table 2: The details of the simulation for single particle sedimentation case

Case	Re_p	St	ρ_f	μ_f
1	1.4	0.19	970	0.373
2	4.1	0.53	965	0.212
3	11.6	1.50	962	0.113
4	31.9	4.13	960	0.058
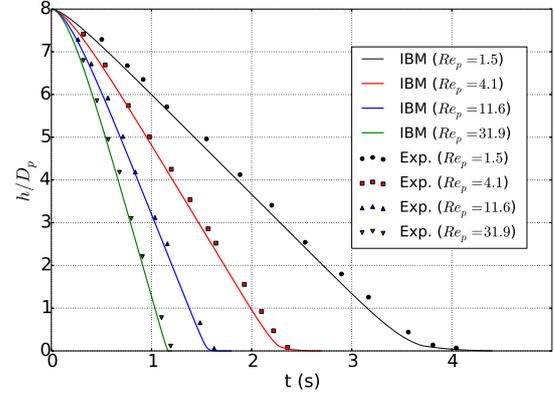
a) Particle location
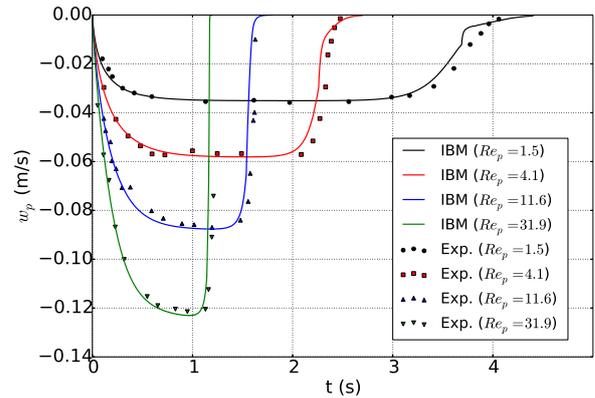
b) Particle velocity

Figure 3: Comparison of numerical simulation with the experimental results for the case of single particle sedimentation

$$Re = \frac{2\rho_f u_f H}{\mu_f}$$

$$Ar = \frac{\rho_f^2 (\rho_p/\rho_f - 1) g D_p^3}{\mu_f^2}$$

The boundary conditions in the x & z directions are periodic and in y-direction, no-slip condition with zero velocity is applied on the upper and lower wall. The pressure gradient is applied in the x-direction to drive the flow.

Effect of initial particle configuration

In this section, the results for the effect of the particle configuration is presented. The simulations are carried out for 2 different cases - **Case 1**: The particles are stacked in three layers with each subsequent layer touching the layer below, **Case 2**: The particles are placed with a gap of $D_p/2$ between each layer. Other relevant simulation parameters are summarized in Table 3.

Table 3: Simulation parameters used to study the effect of initial particle configuration

Simulation parameter	Value
# CV	300 x 160 x 60
# particles	135
Δx	0. 01 m
$D_p/\Delta x$	20
Re	100
Ar	39.24
ε_s	19.6 %

The simulation for both these cases is performed to study the behaviour of particles according to their initial configuration. It is found that for the given cases, the dynamics is substantially different although the non-dimensional parameters are kept constant (Figs. 4 & 5). It can be commented that for Case 2, the spacing between particles allows the particles to move freely due to gravity and lift forces affecting the flow field around it. With the evolution of time, the disturbance in the flow field leads to more pronounced asymmetric forces on the particles and consequently they remain in fluidized state (Fig. 5). On the contrary, no such fluidization is observed for Case 1 and particles remain sedimented during the entire simulation. This might be caused due to the restricted motion of particles in a closed packing. Hence, it is important to have a thin gap of fluid between particles to produce realistic simulations mimicking the proppant transport.
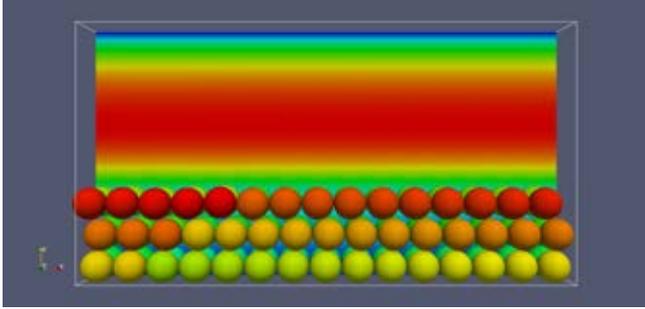
Figure 4: Particle configuration and the velocity distribution (on centre *x-y* plane) at $t^*(tu_f/D_p) = 17$ (Case 1)
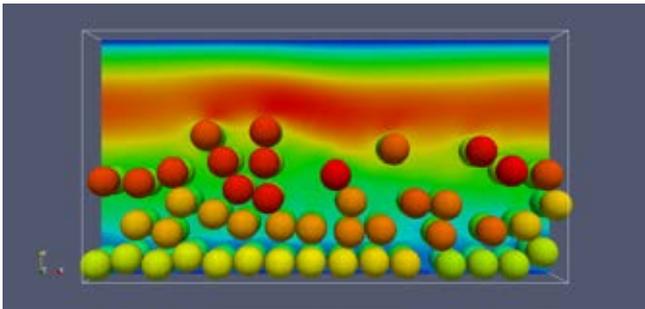
Figure 5: Particle configuration and the velocity distribution (on centre *x-y* plane) at $t^*(tu_f/D_p) = 17$ (Case 2)

Effect of solids volume fraction

In this section, the effect of solids volume fraction on the particle transport and specifically, on the sedimentation times is studied. The initial particle configuration is shown in Fig. 6 whereas Table 4 lists the detailed simulation parameters.
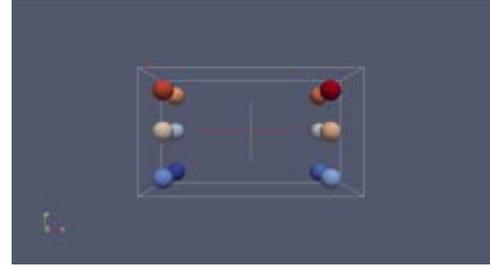
a) $\varepsilon_s = 0.017$
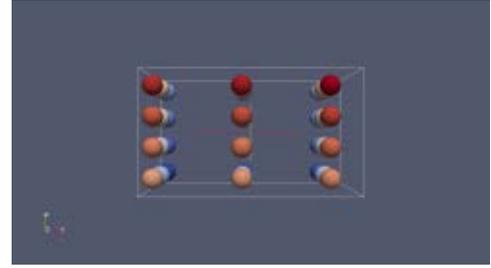
c) $\varepsilon_s = 0.05$
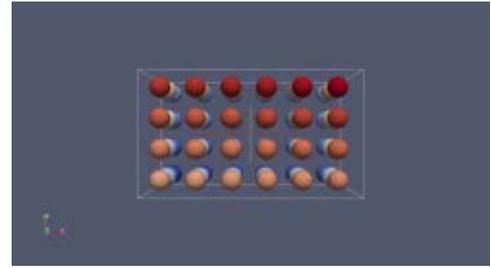
e) $\varepsilon_s = 0.1$
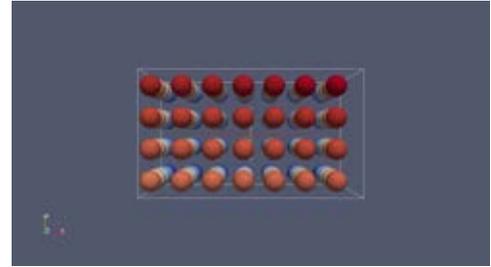
g) $\varepsilon_s = 0.2$

Figure 6: Initial configuration of particles for different solids volume fraction.

In these simulations, the average vertical location (Eq. 13) of all particles is monitored to obtain the sedimentation time. The time at which y_{avg} reaches a steady state value, is used as the sedimentation time and is listed in Table 5 for the chosen parameter space.

$$y_{avg} = \frac{1}{N} \sum_{n=i}^{n=N} y_i \qquad (13)$$

The aim of the fracking process is to attain operating conditions with a longer sedimentation time such that particles are carried deeper into a fracture. It can be observed from the results that the sedimentation time increases by decreasing *Ar* owing to the reduced effect of the gravitation force. However,

Table 4: Simulation parameters used to study the effect of solids volume fraction

Simulation parameter	Value
# CV	200 x 120 x 120
Δx	0. 01 m
$D_p/\Delta x$	20
Re	100 - 400
Ar	0-7848
ε_s	1.7 - 20%

with the higher sedimentation times, it is also important for the particles not to fluidize in the flowback stage of the fracking process. The lighter particles will tend to fluidize quickly in the flowback and the efficiency of the process would be reduced. Hence, it would be important to use the heavier particles and still have a longer sedimentation time. From Table 5, it can be observed that the sedimentation time for $Re = 400$ is comparable for the cases {$\varepsilon_s = 0.017$, $Ar = 1569.6$} and {$\varepsilon_s = 0.2$, $Ar = 7848$} which signifies that the heavier particles can also be transported at a longer distances if the higher solids volume fraction is used. The pattern for the influence of the Reynolds number on the sedimentation time is not consistent for all the volume fractions, however, the Reynolds number tends to increase the sedimentation time at higher value of ε_s.

Ar \ Re	100	200	300	400
0	-	-	-	-
1569.6	13.4	11.2	10.5	9.3
3924	6.7	6.9	6.4	6.1
7848	3.9	4.2	4.7	4.8

a) Sedimentation time (s) ε_s=0.017

Ar \ Re	100	200	300	400
0	-	-	-	-
1569.6	20.2	24.7	23.4	35.2
3924	9.8	10.3	10	12.8
7848	4.7	5.3	6.1	7.4

b) Sedimentation time (s) ε_s=0.05

Ar \ Re	100	200	300	400
0	-	-	-	-
1569.6	21.1	25.6	23.2	18.3
3924	18.4	18.6	18.9	13.2
7848	8.7	9.1	10.2	7.6

c) Sedimentation time (s) ε_s=0.1

Ar \ Re	100	200	300	400
0	-	-	-	-
1569.6	17.2	15.2	22.4	23.2
3924	10.3	13.7	12.8	15.6
7848	8.6	12.6	9.9	11.3

d) Sedimentation time (s) ε_s=0.2

Table 5: Sedimentation time for all cases simulated, various ε_s

CONCLUSION

In this work, an efficient and accurate model for the direct numerical simulation (DNS) of liquid-solid flows is presented. The torque computation results with the improved immersed boundary method (IBM) are presented for the single rotating sphere in a quiescent fluid in comparison with the original IBM. It is found that the improved IBM reduces the error around 3-13 times. The verified and validated IBM is then used to simulate the transport of solid particles in a narrow channel to mimic the proppant transport in rock fractures in a fracking process. Initially, the simulation of transport of 135 particles with two different particle arrangements is performed and it was found that the spacing between the particles leads to a fluidization, contrary to the packed particle system with the same non-dimensional flow parameters. Hence, it is important to have a gap between particles while performing simulations to attain closer similarity with the real process.

As a next step, 48 simulation cases are performed to study the influence of the solids volume fraction, Archimedes number and Reynolds number on the proppant transport phenomenon. It was found that the sedimentation time of the heavier particles can be increased by increasing the solids volume fraction and it is even comparable to the sedimentation time of the lighter particles in the dilute system. Moreover, the influence of increasing Reynolds number is more pronounced and consistent in the higher volume fraction cases and contributes positively to keep the particles fluidized for a longer period.

In reality, the process of proppant transport is quite complex due the rough walls in the rock fractures with the varying widths, visco-elastic fracking fluids, randomly oriented cracks, high aspect ratio in dimensions of a crack and poly-dispersity in the proppant sizes. Moreover, large number of particles are used in this process. Hence, the numerical model has to be extended with the complex boundary conditions for rough walls, visco-elastic flow modeling and poly-dispersity of particles. To simulate the larger system, full parallelization of the fluid solver as well as particle part is also very important.

ACKNOWLEDGEMENTS

–

REFERENCES

BIRD, R.B., STEWART, W.E. and LIGHTFOOT, E.N. (2007). *Transport phenomena*. John Wiley & Sons.

BRENNER, H. (1961). "The slow motion of a sphere through a viscous fluid towards a plane surface". *Chemical Engineering Science*, **16(3-4)**, 242–251.

BREUGEM, W.P. (2012). "A second-order accurate immersed boundary method for fully resolved simulations of particle-laden flows". *Journal of Computational Physics*, **231(13)**, 4469–4498.

CHOI, H.G. and JOSEPH, D.D. (2001). "Fluidization by lift of 300 circular particles in plane poiseuille flow by direct numerical simulation". *Journal of Fluid Mechanics*, **438**, 101–128.

CUNDALL, P.A. and STRACK, O.D. (1979). "A discrete numerical model for granular assemblies". *Geotechnique*, **29(1)**, 47–65.

DAS, S., DEEN, N.G. and KUIPERS, J. (2016). "Immersed boundary method (ibm) based direct numerical simulation of open-cell solid foams: Hydrodynamics". *AIChE Journal*.

DEEN, N.G., KRIEBITZSCH, S.H.L., VAN DER HOEF, M.A. and KUIPERS, J.A.M. (2012). "Direct numerical simulation of flow and heat transfer in dense fluid-particle systems". *Chemical Engineering Science*, **81**, 329–344.

FADLUN, E., VERZICCO, R., ORLANDI, P. and MOHD-YUSOF, J. (2000). "Combined immersed-boundary finite-difference methods for three-dimensional complex flow simulations". *Journal of Computational Physics*, **161(1)**, 35–60.

KEMPE, T. and FRÖHLICH, J. (2012). "An improved immersed boundary method with direct forcing for the simulation of particle laden flows". *Journal of Computational Physics*, **231(9)**, 3663–3684.

PATANKAR, N., KO, T., CHOI, H. and JOSEPH, D. (2001). "A correlation for the lift-off of many particles in plane poiseuille flows of newtonian fluids". *Journal of Fluid Mechanics*, **445**, 55–76.

PESKIN, C.S. (1972). "Flow patterns around heart valves: a numerical method". *Journal of Computational Physics*, **10(2)**, 252–271.

TEN CATE, A., NIEUWSTAD, C., DERKSEN, J. and VAN DEN AKKER, H. (2002). "Particle imaging velocimetry experiments and lattice-boltzmann simulations on a single sphere settling under gravity". *Physics of Fluids*, **14(11)**, 4012–4025.

TSENG, Y.H. and FERZIGER, J.H. (2003). "A ghost-cell immersed boundary method for flow in complex geometry". *Journal of Computational Physics*, **192(2)**, 593–623.

UHLMANN, M. (2005). "An immersed boundary method with direct forcing for the simulation of particulate flows". *Journal of Computational Physics*, **209(2)**, 448–476.