Numerical simulation of mono-disperse gravity-driven granular flow around a wedge using two-fluid model

Georgy Shoev
Khristianovich Institute of Theoretical and Applied Mechanics, 630090, 4/1 Institutskaya str., Novosibirsk, Russia
E-mail: shoev@itam.nsc.ru

Abstract. The numerical simulation of glass beads granular flow around a wedge located in a rectangular channel is performed by the Two-Fluid Model (i.e. the Euler-Euler approach) aiming to make a comparison between the numerical and the available experimental data. The channel is tilted against the horizon making the glass beads fall under the action of the gravity force. The numerical simulation shows the glass beads structure that resembles a bow shock wave around the wedge. The computed bow shock stand-off distance is compared to the data extracted from the experimental results. The details of the comparison between the numerical and the experimental data are discussed.

1. Introduction
Granular flows are of high importance for different industrial applications. Accurate prediction of a granular flow behaviour is necessary for research and development studies. The Euler-Euler or Two-Fluid Model (TFM) is one of the possible approaches that can be used for the numerical simulation of different multiphase flows. TFM was tested to describe the multiphase flow in fluidized beds, where solid particles hover due to the air jets directed against the vector of the gravity force. Usually experimental data [1, 2] show the flow parameters averaged in space and time. These averaged experimental results compare almost equally to the results of the numerical simulation based on different modifications of TFM (different drag models) even though the instantaneous flow fields can be significantly different [3].

The granular flow of solid particles and the flow of gas behave similar to a certain extent. One of such similarities is the structure that resembles the shock wave in classical gas dynamics. These shock waves in granular flows [4, 5, 6] can be observed around different bodies such as a cylinder, a square, a wedge, etc. In these cases a granular flow reaches a steady state for some period of time allowing the detailed analysis of the flow features like a detached bow shock wave, shear instabilities and a granular wake (a region of the low volume fraction of particles behind the body). These external granular flows are similar to high-speed gas flows (for example, [7, 8] and many others).

One of the classical flow problems in gas dynamics is a supersonic gas flow around a wedge. The study of this flow problem shows many important mechanisms of high-speed gas flows. Just as in classical gas dynamics, the investigation of an external granular flow around a
wedge can reveal many important fundamental mechanisms, and therefore, this flow problem
attracted many researchers [4, 5, 6] to study it theoretically, numerically and experimentally.
The experimental results [4] for a granular flow around a wedge can be used to validate
different numerical models. The important thing is that this flow reaches a steady state,
here, the necessity of averaging over time is eliminated. The aim of this study is to assess
the capability of the two-fluid model implemented in the ANSYS Fluent software to describe
a mono-disperse gravity-driven granular flow around a wedge for the conditions of the recently
performed experiments [4].

2. Problem formulation and numerical procedure
The current numerical setup consists of a reservoir and a fully transparent channel, where the
wedge is located (figure 1). Four cases with different values of a half-angle of the wedge \( \theta \)
are considered, namely, 50\(^\circ\), 60\(^\circ\), 70\(^\circ\) and 80\(^\circ\). The reservoir and the channel are tilted against the
horizon by \( \phi = 50^\circ \) angle. At the initial moment, the computational domain has the volume
fraction of the glass beads \( \varepsilon_x = 0.55 \). The glass beads have the diameter of \( d = 125 \mu m \) and the
density of \( \rho = 1600 \) kg/m\(^3\). The computation is performed using transient (unsteady) mode.
Moving down the channel the glass beads interact with the wedge forming a steady state flow
in several seconds. Convergence to the steady state was controlled by the independence of the
flowfield of the glass beads volume fraction \( \varepsilon_x \) on the front \((z = 5 \) mm\) and the back \((z = 0 \) \)
walls and in the middle plane \((z = 2.5 \) mm\) of the channel. Additionally, the profile of the glass
beads volume fraction along the stagnation line was monitored to make sure that the numerical
solution is time independent.

The air flow is assumed to be laminar and the air is considered a single species gas with
a constant density (incompressible flow). The solid particles and the air gas are considered
inert media without any chemical reaction, although, the Euler-Euler approximation allows
taking into account the chemical interaction between the carrier and the disperse phases [9, 10].
The energy equation is not included in the modelling. The coefficient of dynamic viscosity
is assumed to be constant under normal conditions. The glass beads are considered through
the Eulerian phase. The granular temperature model uses the phase property option of the
flow solver. The following solver settings were used for the glass beads phase simulation:
Gidaspov for the granular viscosity, lun-et-al for the granular bulk viscosity, schaeffer for
the frictional viscosity. Angle of internal friction was set to 30\(^\circ\), based-ktgf option was used

Figure 1. Computational domain and boundary conditions.

Figure 2. Sub-domains for the computational mesh.

Figure 3. Volume fraction of glass beads along line \((x, 0, 2.5 \) mm\) for different computational meshes.
the frictional pressure, derived for the frictional modulus, the friction packing limit was equal to 0.63, the granular temperature used algebraic option, lun-et-al was used for the solids pressure computation, ma-ahmadi for the radial distribution, derived for the elasticity modulus. The packing limit was set to 0.99. The Gidaspow drag model [11] is used to describe phases interaction, i.e. the carrier phase affects the disperse phase and vice versa (note: one way coupling, like it was done in [12], cannot be applied). The other options for phases interaction are the lift coefficient, set to none, the restitution coefficient set to 0.97, and the surface tension coefficient set to none.

The computational domain contains three types of boundary conditions: pressure-outlet, wall and symmetry (figure 1). “Pressure outlet 1” is located at the top of the reservoir. During the computation the glass beads are falling down because of gravity causing reversed flow into the computational domain, hence, the volume fraction of the glass beads has to be specified and it is equal to \( \varepsilon_s = 0.55 \). The real size of the reservoir does not seem to be important because the mass flux into the channel seems to be independent on the quantity of the granular material in the reservoir as it was mentioned in [4]. “Pressure outlet 2” is located downstream of the wedge assuming that the flow behind this boundary does not affect the flow structure around the wedge. Both pressure-outlets use atmospheric pressure. In order to reduce the computational cost, a symmetry plane at \( y = 0 \) is used. The other boundaries have the type of wall, where zero shear stress is used (the inviscid wall).

The computational domain is divided into several sub-domains in order to build the computational mesh consisting of hexahedral cells. The number of computational cells on each edge of the sub-domains is given in figure 2, where \((x,y)\) plane is shown. The computational mesh is refined toward the surface of the wedge in order to capture shock-like structures of the glass beads. The height of the cell adjacent to the wedge surface is about 0.17 mm while the farthest cell from the wedge surface in the sub-domain is about 2.9 mm. The number of cells in \( z \)-direction is varied because the thickness of the glass beads layer changes and can cause high gradients of the particles volume fraction. The following numbers of cells in \( z \)-direction inside the glass channel is used: 4 (Mesh 1), 8 (Mesh 2), 16 (Mesh 3) and 24 (Mesh 4).

Phase coupled SIMPLE algorithm is used for the pressure-velocity coupling. The spatial discretization uses first order upwind option for the momentum, volume, and granular temperature and least square cell based for gradients. Option for the transient formulation was set to first order upwind. Default values of the under relaxation factors are used for mesh 1 and 2, while they are two times reduced for meshes 3 and 4. Dual time stepping is used with 30 iterations per one step which is fixed at 0.001 sec.

Figure 3 shows the volume fraction of the glass beads along the line \((x, 0, 2.5 \text{ mm})\) for different computational meshes. These profiles include the reservoir and the part of the glass channel upstream the wedge. Close to the point \( x = 0 \) a rapid increase of the glass beads volume fraction is observed indicating an appearing bow shock wave. As can be seen, the grid convergence is not reached, but the stand-off distance of the bow shock is not very sensitive to the spatial resolution.

3. Results

Figure 4 shows the profiles of the glass beads volume fraction along different lines \((x, 0, 0), (x, 0, 2.5 \text{ mm})\) and \((x, 0, 5 \text{ mm})\) ahead of the wedge with \( \theta = 60^\circ \). As we can see, the volume fraction of particles depends strongly on \( z \)-coordinate because of gravity that drags the glass beads to \( z = 0 \). Moreover, the volume fraction profile changes in the free-stream upstream of the bow shock, which is also caused by the influence of gravity. It is worth noting that the glass beads volume fraction exceeds the maximum packing limit (\( \sim 0.63 \)) on the glass channel bottom \((z = 0)\) in the free-stream as well as in the close vicinity of the wedge apex. The profiles at the middle \((z = 2.5 \text{ mm})\) and at the top \((z = 5 \text{ mm})\) show that the glass beads volume fraction
exceeds the maximum packing limit only in the bow shock or behind it close to $x = 0$. It is worth noting that the flow solver is not particularly fit for the accurate simulation of complicated gas-solid interactions. On the other hand, the flow solver has already been successfully applied to many specific flow problems [13, 14, 15, 16].

**Figure 4.** Volume fraction of glass beads at $y = 0$ for different $z$-coordinates.

**Figure 5.** $z$-velocity ($w_z$) of glass beads at $y = 0$ for different $z$-coordinates.

**Figure 6.** Average volume fraction of glass beads at $y = 0$.

**Figure 7.** Shock stand-off distance: experimental data [4] and numerical data.

Figure 5 shows $z$-velocity component, $w_z$, of the glass beads. As we can see the particles at $z = 1$ mm rise in the bow shock and form a flat, plateau-like velocity profile behind it. In the middle ($z = 2.5$ mm) and close to the top ($z = 4$ mm) of the channel, the particles have more complicated behaviour of $w_z$ that includes regions with negative and positive values.

Figure 6 shows the particles volume fraction averaged in $z$-direction. In a certain sense, the volume fraction averaged across the channel corresponds to the experimental measurements [4]. At the same time, the experimental shadowgraph images show only light coloured (low volume fraction) and dark coloured (high volume fraction) regions, and they do not provide information about the absolute value of the volume fraction. Therefore, the accurate comparison between experimental and numerical data is quite a challenging task. Here, the first step of such comparison is presented in figure 7 for the bow shock stand-off distance. Numerical stand-off distances are taken from the profiles of the averaged volume fraction given in figure 6. Since the free-stream value of the averaged volume fraction is not constant it is hard to identify the
beginning of the bow shock. Therefore, the first peak of the averaged volume fraction of the
glass beads is taken as the position to determine the numerical stand-off distance of the bow
shock. Although this criterion seems to be straightforward, an agreement between the numerical
and the experimental data in figure 7 is not observed. The accurate experimental identification
of the averaged value of the glass beads volume fraction in the beginning of the bow shock can
make such comparison more correct. On the other hand, it can be noticed that the numerically
predicted averaged value of the glass beads volume fraction is much higher than the maximum
packing limit ($\sim 0.63$). This problem of TFM computation also needs to be properly resolved in
the future in order to make a correct validation of the numerical model.

4. Conclusion
The Two-Fluid Model implemented in ANSYS Fluent can qualitatively reproduce the structure
of a gravity-driven granular flow around a wedge, although the considered model does not
accurately quantify some of the flow features:

- at the bottom of the channel ($z = 0$) the volume fraction of the glass beads ($\varepsilon_s \sim 0.9$)
exceeds the maximum packing limit ($\sim 0.63$);
- in the close vicinity of the wedge apex the volume fraction of the glass beads also increases
higher than the maximum packing limit.

The comparison between the numerical and the experimental results shows a disagreement
that can be explained by:

- inaccuracy of the model for calculating the volume fraction of the glass beads;
- impossibility of the accurate comparison of the bow shock stand-off distance in the
experiment and the calculation, since the experimental data do not allow reading the
absolute value of the volume fraction of the glass beads averaged through the glass channel.

Acknowledgments
The author would like to thank Ronith Stanly for the first discussion of this work, Aqib Khan
for the clarification of the experimental setup. The author is especially grateful to Anna Shoeva,
who helped to proof-read the paper. The research was partly carried out within the framework of
the Program of Fundamental Scientific Research of the state academies of sciences in 2013-2020
(project No. AAAA-A17-117030610138-7) and Russian Foundation for Basic Research (project
No. 18-38-20113). Computations are performed at Novosibirsk State University (NSU).

References
[1] Kuipers J 1990 Ph.D. thesis University of Twente
[2] Muller C, Holland D, Sederman A, Scott S, Dennis J and Gladden L 2008 Powder Technology 184 241–253
[3] Stanly R and Shoev G 2018 Chemical Engineering Science 188 132–149, doi:10.1016/j.ces.2018.05.030
[4] Khan A, Hankare P, Verma S, Kumar R and Kumar S 2019 32nd International Symposium on Shock Waves
[5] Gray J and Cui X 2007 J. Fluid Mech. 579 113–136, doi:10.1017/S0022112007004843
[6] Garai P, Verma S and Kumar S 2019 J. Vis. 22 729–739, doi:10.1007/s12650–019–00558–5
[7] Bondar Y, Markelov G, Gimelshein S and Ivanov M 2006 J. Thermophys. Heat Tr. 20 699–709, doi:10.2514/1.18758
[8] Bountin D, Gronyko Y, Kirilovskiy S, Maskov A and Poplavskaya T 2018 Thermophys. Aeromech. 25 483–
496, doi:10.1134/S0869864318040029
[9] Kratova Y, Kashkovsky A and Shershnev A 2018 AIP Conference Proceedings 2027 040086, doi:10.1063/1.5065360
[10] Kratova Y, Kashkovsky A and Shershnev A 2019 Thermal Science 23 623–630, doi:10.2298/TSCI19S2623K
[11] Gidaspow D 1994 Multiphase Flow and Fluidization ed Gidaspow D (San Diego: Academic Press)
[12] Kudryavtsev A, Shershnev A and Ryblyova O 2019 International Journal of Multiphase Flow 114 207–218, 
doi:10.1016/j.ijmultiphaseflow.2019.03.009
[13] Polivanov P, Gromyko Y, Sidorenko A and Maslov A 2017 *J. Appl. Mech. Tech. Phy.* **58** 845–852, doi:10.1134/S0021894417050108

[14] Polivanov P 2018 *Thermophys. Aeromech.* **25** 789–792, doi:10.1134/S0869864318050153

[15] Mironov S G, Poplavskaya T V and Kirilovskiy S V 2017 *Thermophys. Aeromech.* **24** 629–632, doi:10.1134/S0869864317040151

[16] Kirilovskiy S, Poplavskaya T, Tsyryulnikov I and Maslov A 2017 *Thermophys. Aeromech.* **24** 421–430, doi:10.1134/S0869864317030106