Multidimensional Gas Dynamics Simulation of the Attenuation of Shock Wave by the Granular Beds

Sidorenko D., Utkin P.*, Maksimov F.

Institute for Computer Aided Design RAS, Moscow, Russia
**Corresponding author's email: pavel_utk@mail.ru*

ABSTRACT

We consider the problem of interaction of the shock wave with a system of non-moving cylinders that model the granular media bed of the particles in the initial stage of the interaction. The problem is solved using both Euler and Navier-Stokes systems of equations. The numerical algorithm for the solution of Euler equations is based on the Cartesian grid method, for Navier-Stokes equations – on the multiblock computational technology. Numerical experiments are conducted using both numerical approaches.

KEYWORDS: Numerical simulation, shock wave, Cartesian grid, multiblock technology.

NOMENCLATURE

D	particle diameter (m)
H	channel height (m)
L	channel length (m)
l	length of the particles bed (m)

Greek

θ	dimensionless parameter (-)
ε	porosity (-)

INTRODUCTION

The problem about the shock wave (SW) – particles cloud interaction is the fundamental basis for a whole class of problems involving the flows of dense two-phase media with SWs. Such problems directly concern safety applications in mining and industry. Examples are the weakening of blast waves by the granular beds and the dust dispersion in the initial stages of dust explosions. Early studies first dealt with the small volume fractions of the particles in the cloud. In fact, the experiments related to dusty gas flows. In [1], the clouds with a volume fraction of particles 0.1 – 3% were considered. It was shown, both experimentally and numerically, the effect of qualitative change in the supersonic flow behind the SW in a cloud. There are a much smaller number of experimental studies for the case of dense clouds [2, 3]. The last studies in this field [4] show the unusual result that, for the case of micro-particles, the relationship of the shock attenuation has a negative correlation with the dimensionless complex [5]:

$$\theta = f_m \frac{(1-\varepsilon)l}{\varepsilon D}, \qquad (1)$$

where f_m is the pressure drop coefficient, ε is the void fraction of the particle wall, l is the total length of the densely packed particles, and D is the diameter of the individual particle.

Numerical studies of the problem began on a new level several years ago. In the field of dense flows of two-phase media with SW new models based on the kinetic theory of granular media were

Proceedings of the Ninth International Seminar on Fire and Explosion Hazards (ISFEH9), pp. 308-313
Edited by Snegirev A., Liu N.A., Tamanini F., Bradley D., Molkov V., and Chaumeix N.
Published by St. Petersburg Polytechnic University Press
ISBN: 978-5-7422-6496-5 DOI: 10.18720/spbpu/2/k19-41

constructed [6, 7]. In the field of Lagrangian methods the discrete element method was developed that takes into account the volume fraction of the particles [8]. However the models [6 – 8] demand the use of closing relationships, in particular for the description of the interphase interactions that are traditionally are taken from ~~the~~ natural experiments.

At the same time, it has recently become possible to simulate the interaction of the SW with the assembly of particles at the "micro level", or by using the direct numerical simulation of the interaction of the flow with each particle [9 – 11]. In our previous study [12] we investigated the problem of SW – system of cylinders interaction in a statement correspondent to the natural experiments [5] about SW – particles bed interaction using Euler equations. The goal of this work is the comparative analysis of the simulations of weakening of the SW due to its interaction with the system of cylinders, using both Euler and Navier-Stokes (NS) equations.

STATEMENT OF THE PROBLEM

The statement of the problem follows the experimental study in [5]. Consider a plane channel of length L = 88.4 cm and height H = 1.7 cm (see Fig. 1). There are 6 non-moving rigid cylinders of diameter D with the centers at the fixed points C_1(40.3; 0.85), C_2(41.8; 0.0), C_3(41.8; 1.7), C_4 (43.2; 0.85), C_5(44.7; 0.0), C_6(44.7; 1.7) (all sizes are in cm). The following diameters are considered: 0.74, 0.85, 1, 1.2, 1.36, and 1.48 cm. The incident plane SW with the Mach number 1.47 is located at the point x_{SW} = 35.7 cm, some distance to the left from the system of bodies. The parameters of the gas in front of the SW are normal. We consider not the whole channel height as in [5], but only a thin part of it, with slip wall boundary conditions at the top and bottom boundaries. The conditions at the boundaries of the cylinders are slip walls in Euler simulations, and no slip walls in NS simulations. The inflow condition is imposed at the left boundary and the slip wall at the right. In the computation time, the transmitted and reflected waves have no time to reach the right and left domain boundaries.

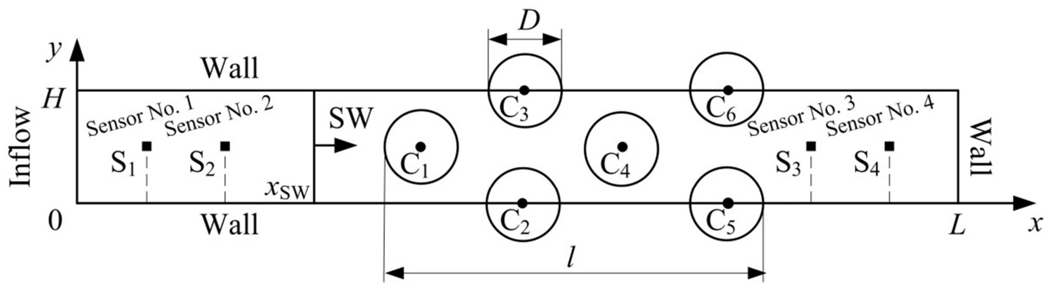

Fig. 1. Schematic statement of the problem.

The objects of the investigation are the reflected and transmitted waves that are formed as a result of the interaction of the incident SW with the cylinders. The intensities of the waves are determined with the use of four pressure transducers S_1 – S_4.

MATHEMATICAL MODELS AND NUMERICAL ALGORITMS

The Eulerian approach

Two mathematical models were considered. The first one is based on two-dimensional Euler equations. These are solved in the computational domain obtained by the subtraction of the cylinders from the rectangular area. The numerical algorithm is based on the Cartesian grid method and is described in detail in [13]. The main idea of the method is the usage of the computational grids with the square cells, and the special consideration of the small cells obtained as a result of the intersection of the regular square cells with the curvilinear boundaries of the bodies, using so called "h-boxes " [14].

The numerical algorithm and its program realization were tested on a number of SW problems. For the problem of regular and simple Mach reflections of the SW from the wedge, the comparison of the simulated distribution of the density along the wedge with the experimental data [15] gives a maximal error of 8%. In the problem of SW – cylinder interaction, the pressure distribution along the surface during SW diffraction coincides with the simulations using the detailed Navier-Stokes equations [16] in the range of 1%. We also considered the problem of the interaction of SW with Mach number 1.4, with the system of regularly arranged 16 cylinders [17]. The derived locations of the pressure peaks at the upstream and downstream sensors, in comparison with [17], differ by not more than 5%, the amplitudes – not greater than 10%.

The Navier-Stokes approach

For the solution of NS equations, multiblock computational technology is used. A grid is constructed from a set of several structured grids. The first basic grid *I* consists of rectangular cells (see Fig. 2). It is intended for computing the outer inviscid flow and has no relation to the body. The second grid *II* is adapted to the body surface. Specifically, the lines of one of the coordinate families of this grid are directed normally to the body. The nodes along these coordinate lines are condensed exponentially toward the solid surface. To match the solutions obtained on grids *I* and *II*, after performing an integration step, the gasdynamic functions on the outer boundary *L* of grid *II* are determined by interpolating the solution obtained on grid *I*. At the same time, the solution at all nodes of *I* that belong to the domain covered by *II* is replaced by the solution obtained on grid *II*. A two-step difference scheme is used.

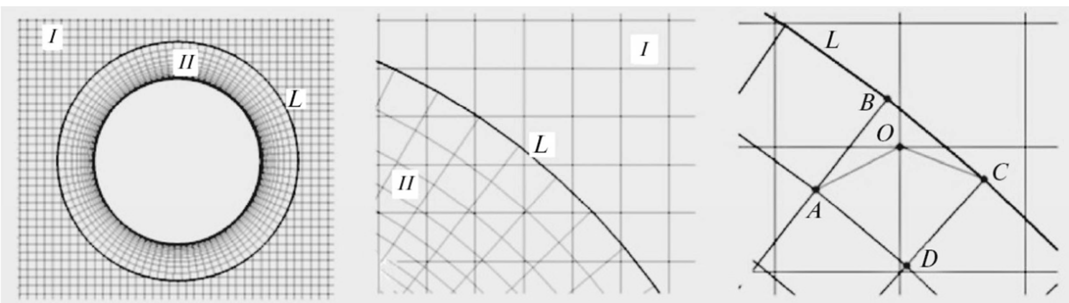

Fig. 2. The multiblock computational technology.

Interpolation is used to recalculate the gasdynamic functions from one grid to the other. In the considered two-dimensional case, the interpolation procedure is as follows. Whether the point O belongs to the cell $ABCD$ is checked by enumeration. This problem is solved only once to determine interpolation coefficients, so there is no need to use faster cell-searching methods. The values of functions at the node O are determined in terms of their values at the nodes A, B, C, and D. The particular computation method to be used depends on the physical features of the problem under study. For example, the value of f at the node O can be determined in terms of its values at any three nodes (say B, A, and D) by applying the interpolation formula $f_O = f_A + \alpha(f_B - f_A) + \beta(f_D - f_A)$, where $\alpha = |AO \times AB|\ |AD \times AB|$ and $\beta = |AO \times AD|\ |AB \times AD|$. To take into account the function value at the node C, in a similar manner, we can express f_O in terms of its values at the points of another triplet, for example, D, C, and B. The final expression for f_O is the arithmetic mean of the values produced by two indicated methods.

Interpolation coefficients can be chosen using mathematical considerations (including an increase in the number of interpolation nodes to ensure the required order of accuracy), or physical considerations. For example, by analyzing the direction of the velocity at cell nodes. The node *A* can be specified as one at which the velocity is directed toward the point *O* (which is reasonable in the case of supersonic flows). In various problems, it is reasonable to choose interpolation coefficients, by applying the most suitable of the above-indicated methods. The described multiblock technology was used for the solution of a number of the problems of the supersonic aerodynamics [18].

Both numerical algorithms for Euler equations and for NS equations have the second approximation order.

RESULTS OF SIMULATIONS

Euler simulations are carried out at the uniform grid 10,400×200. The uniform grid for the inviscid part of the NS simulations contains 18,000×400 cells. The grids adapted to the cylinders surfaces contains 185×40 cells, with the exponential distribution of the cell size in the radial direction. The resolutions were taken from the convergence study. The difference in the resolution of the grids in Euler and NS simulations is due to the more diffusive numerical flux in NS simulations (MacCormack instead of Godunov in Euler simulations). NS simulations are carried out for a Reynolds number 10^5.

The general character of the process of the SW – system of cylinders interaction is as follows. At the initial stage of the process of SW – array of cylinders interaction, the reflected wave consists of several parts, due to the incident SW reflection from the single cylinders forms. The strength of the reflected wave increases in time because of the additional contribution of the disturbances caused by the interaction of the leading SW with the cylinders downstream. As a result, after some time, the almost flat collective reflected wave and transmitted wave forms. The flow in the wake of the "cloud" is highly perturbed, with the formation of the structures like Karman vortex sheet.

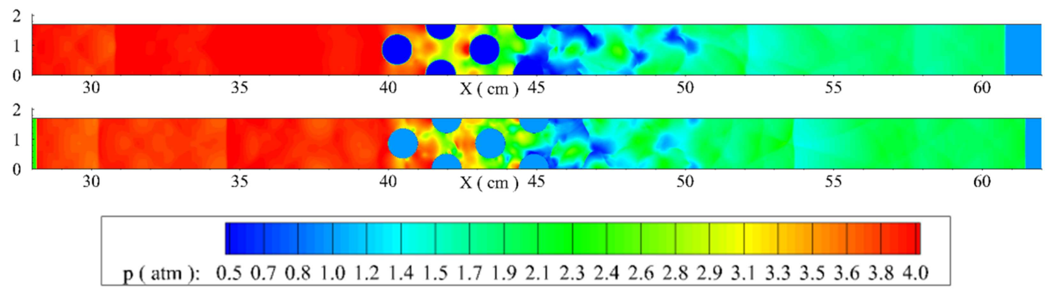

Fig. 3. Predicted pressure field in Euler simulation (upper) and NS simulation (lower) at the same time instant 0.4 ms. D = 1 cm. The scale is in atmospheres.

Figure 3 illustrates the typical spatial distributions of pressure in the numerical experiments. In the Euler simulation, the reflected wave is stronger and faster than in the NS simulation. For the transmitted wave, the situation is the opposite. The flow pattern near the front cylinders is similar in both cases, but for the back cylinders that are in the wake of the front ones, the flow patterns differ significantly. In general in the viscous simulation the pressure in the back part of the system of cylinders is higher than in the non-viscid one. Fig. 4 illustrates typical pressure curves. Sensors Nos. 1 and 2 register~~s~~ the reflected wave. The closest for the bed sensor No. 2 shows that, due to the multiple reflections of leading SW from the cylinders, the pressure reaches the average level after about 0.3 ms, after the beginning of the interaction. Sensors No. 3 and 4 register the transmitted wave. For the sensors, pressure oscillates near the average value because of the vortex structures in the wave behind the system of bodies. By the averaging of such types of pressure curves, the average pressure for each sensor were measured. Fig. 5 generalizes the obtained results for different diameters of the cylinders.

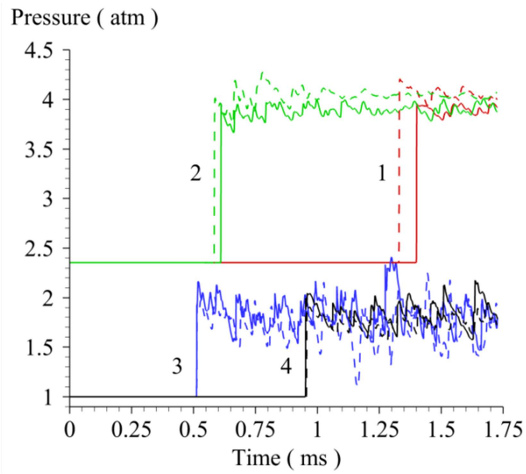

Fig. 4. Predicted pressure curves at the sensor No. 1 (red), No. 2 (green), No. 3 (blue) and No. 4 (black) in Euler simulation (dashed) and NS simulation (solid). *D* = 1 cm.

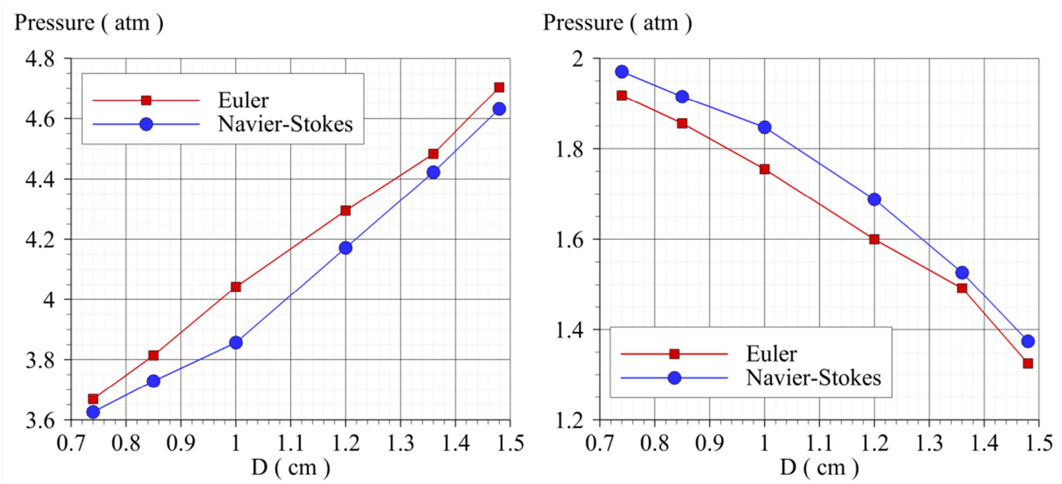

Fig. 5. Average pressure at the sensor No. 2 (left) and sensor No. 4 (right) for different *D*. Red – Euler simulation, blue – NS simulation.

CONCLUSIONS

Comparative analyses of the simulations of the shock wave – system of cylinders interaction using Euler and Navier-Stokes equations have been carried out. At distances of several diameters from the cylinder system, the differences between the simulations using Euler and Navier-Stokes equations are minimal. The average pressures behind the collective reflected and transmitted waves at distances of several tens of diameters are compared. In the Euler simulation the reflected wave is stronger and faster than in the Navier-Stokes simulation. For the transmitted wave, the situation is the opposite. The effect is about 5%. So the Euler simulations are preferable at the distances of several diameters of the cylinder. In addition, the Euler simulations are much more economical in CPU time. At a distance of several tens of diameters the viscous effects in the Navier-Stokes simulations can refine the results.

REFERENCES

[1] V.M. Boiko, V.P. Kiselev, S.P. Kiselev, A.N. Papyrin, S.V. Poplavsky, V.M. Fomin, Shock Wave Interaction with a Cloud of Particles, Shock Waves 7 (1997) 275–285.

[2] X. Rogue, G. Rodriguez, J.F. Haas, R. Saurel, Experimental and Numerical Investigation of the Shock-Induced Fluidization of a Particles Bed, Shock Waves 8 (1998) 29–45.

[3] J.L. Wagner, S.J. Beresh, S.P. Kearney, W.M. Trott, J.N. Castaneda, B.O. Pruett, M.R. Baer, A Multiphase Shock Tube for Shock Wave Interactions with Dense Particle Fields, Exp. Fluids 52 (2012) 1507–1517.

[4] H. Lv, Z. Wang, Y. Zhang, J. Li, Shock Attenuation by Densely Packed Micro-Particle Wall, Exp. Fluids 59 (2018) 140.

[5] S.P. Medvedev, S.M. Frolov, B.E. Gel'fand, Attenuation of Shock Waves by the Beds of Granular Materials, J. Phys. Eng. 58 (1990) 924–928.

[6] T. Khmel, A. Fedorov, Numerical Simulation of Dust Dispersion using Molecular-Kinetic Model for Description of Particle-to-Particle Collisions, J. Loss Prevent. 36 (2015) 223–229.

[7] R.W. Houim, E.S. Oran, A Multiphase Model for Compressible Granular-Gaseous Flows: Formulation and Initial Tests, J. Fluid Mech. 789 (2016) P. 166–220.

[8] A. Kimura, A. Matsuo, Numerical Investigation of the Gas-particle Flow in the Shock Tube Using Discrete Particle and Continuum Model, AIAA Aviation Forum, 2018 Fluid Dynamics Conference, 9 p.

[9] J.D. Regele, J. Rabinovitch, T. Colonius, G. Blanquart, Unsteady Effects in Dense, High Speed, Particle Laden Flows, Int. J. Multiphase Flows 61 (2014) 1–13.

[10] I.A. Bedarev, A.V. Fedorov, Direct Simulation of the Relaxation of Several Particles behind Transmitted Shock Waves, J. Eng. Phys. Thermophys. 90 (2017) P. 423–429.

[11] Y. Mehta, C. Neal, K. Salari, T.L. Jackson, S. Balachandar, S. Thakur, Propagation of a Strong Shock Over a Random Bed of Spherical Particles, J. Fluid Mech. 839 (2018) P. 157–197.

[12] D.A. Sidorenko, P.S. Utkin, Two-Dimensional Gas Dynamics Modeling of the Interaction of a Shock Wave with the Beds of Granular Media, Russian J. Phys. Chem. B 12 (2018) 869–874.

[13] D.A. Sidorenko, P.S. Utkin, A Cartesian Grid Method for the Numerical Modeling of Shock Wave Propagation in Domains of Complex Shape, Num. Meth. Programm. 17 (2016) 353–364.

[14] M. Berger, C. Helzel, A Simplified H-Box Method for Embedded Boundary Grids, SIAM J. Sci. Comput. 34 (2012) A861–A888.

[15] H.M. Glaz, P. Colella, I.I. Glass, R.L. Deschambault, A Numerical Study of Oblique Shock-Waves Reflections with Experimental Comparisons, Proc. Royal Soc. London A. 398 (1985) 117–140.

[16] D. Drikakis, D. Ofengeim, E. Timofeev, P. Voionovich, Comparison of Non-Stationary Shock Wave/Cylinder Interaction Using Adaptive-Grid Methods, J. Fluid Struct. 11 (1997) 665–692.

[17] A. Chaudhuri, A. Hadjadj, O. Sadot, E. Glazer, Computational Study of Shock-Wave Interaction with Solid Obstacles Using Immersed Boundary Methods, Int. J. Num. Meth. Eng. 89 (2012) 975–990.

[18] S.V. Guvernyuk, F.A. Maksimov, Supersonic Flow Past a Flat Lattice of Cylindrical Rods, Comp. Math. Math. Phys. 56 (2016) 1012–1019.