Experimental and numerical research on heat and air flow through a granular material

E Szymanek¹, A Tyliszczak¹
1 Faculty of Mechanical Engineering and Computer Science, Czestochowa University of Technology. Al. Armii Krajowej 21, 42-201 Czestochowa, Poland
E-mail: ewaszym@imc.pcz.pl, atyl@imc.pcz.pl

Abstract. The paper is devoted to experimental and CFD (Computational Fluid Dynamics) studies of heat and mass transfer processes in granular material layers. The 3D numerical model is based on a high-order computational code combined with the immersed boundary method (IB) for modelling the flow and temperature distributions in geometrically complex systems. The temperature measurements are performed using thermocouples placed inside the layers closed in a climatic chamber. Besides providing data for validation of the numerical results one of the key tasks of the experiment was to examine an impact of an external temperature changes on the temperature distribution in the interior of the layers depending on their structures and physical properties. The obtained results reveal very complex flow and heat transfer inside the layers. It is shown that the temperature inside the layers can be effectively changed and controlled by the parameters of air flowing through them.

1. Introduction
The research related to the flow and heat transfer in granular materials is motivated by a wide range of its engineering and industrial applications, e.g. in the fermentation process, for storage of grains, in systems for production and recovery of crude oil, in fluidized beds, in conversion and thermal storage systems (catalytic reactors, energy storage systems). These various applications enforce the need for better understanding of the basics of transport phenomena in granular layers. Their structures are complicated and specification of the configuration of the material’s arrangement inside them is impossible in real conditions. The configurations of the layers for the needs of mathematical and numerical models are usually idealised. Although such descriptions are fraught with errors and will not accurately reflect real structures of the layer they enable analysis of the phenomena inside the layers using discrete/finite elements type methods [1, 2, 3, 4] or continuous and fractional models [6]. These approaches and models are widely used by many researchers but they are not sufficient to accurately describe all processes in granular layers.

In this paper the flow and heat transfer was modelled using the immersed boundary technique based on a volume penalization method (IB-VP)[7, 8], which allows the use of Cartesian meshes for objects with very complex shapes. Although, in case of the granular layers this would translate to arbitrary form of granulates in the present studies the layers were modelled in a simplified manner assuming them as regions of packed spheres. The numerical code used in this research was an in-house code called SAILOR. It is based on the unsteady equation of heat conduction (3D) and the Navier-Stokes equations. The analysed problem was treated as unsteady and the flow was assumed to be laminar. The experimental research consisted
on temperature measurements and was performed using LTCL600 Series 3 climatic chamber (TASLtd., WestSussex, Great Britain). The purpose of the experiment was to determine internal temperature distributions and their variability in respect to a changing temperature outside the layers. These results were then used also as the exemplary data to validate the numerical solutions. We focused on the flow behaviour inside the granular layers and analysed their evolution in between the granulates. We investigated how the flow movement changes the temperature distribution inside the layers.

2. Mathematical modelling

Variable density, variable temperature low Mach number flows are described by the continuity equation, the Navier-Stokes equations and the energy equation defined as:

\[ \frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \] (1)

\[ \rho \left( \frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} \right) + \nabla p = \nabla \cdot \mathbf{\tau} + \mathbf{f}^{IB} \] (2)

\[ \rho C_p \left( \frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T \right) = \nabla \cdot (\kappa \nabla T) + \mathbf{f}_T^{IB} \] (3)

where \( \rho \) stands for the density, \( T \) - temperature, \( p \) - hydrodynamic pressure, \( \mathbf{u} \) - velocity vector, \( C_p \) - heat capacity and \( \kappa \) - heat conductivity. The set of Eqs. (1)-(3) is complemented with the equation of state \( p_0 = \rho R T \), where \( p_0 \) denotes the thermodynamic pressure and \( R \) - is the specific gas constant. In open flows with inlet/outlet boundaries \( p_0 \) is constant in space and time [14] and in this work it is assumed to be 101325Pa. The molecular viscosity (\( \mu \)) within the viscous stress tensor \( \mathbf{\tau} \) is computed from the Sutherland law.

IB-VP source term. The source terms \( \mathbf{f}^{IB} \) and \( \mathbf{f}_T^{IB} \) originate from the immersed boundary volume penalization approach (IB-VP). Their role is to penalize a difference between the actual and assumed velocity and temperature of the solid body. They act on a fluid in such a way as if there were solid objects immersed in the flow domain and they are defined as:

\[ \mathbf{f}^{IB} = -\frac{\rho}{\eta} \Gamma(x)(\mathbf{u} - \mathbf{u}_s), \quad \mathbf{f}_T^{IB} = -\frac{\rho C_p}{\eta} \Gamma(x)(T - T_s) \] (4)

where \( \eta \ll 1 \) is the so-called penalization parameter with dimension of time unit and \( \Gamma \) - the phase indicator defined as:

\[ \Gamma(x) = \begin{cases} 0, & \text{for } x \in \Omega_f \\ 1, & \text{for } x \in \Omega_s \end{cases} \] (5)

where \( \Omega_f \) and \( \Omega_s \) are the regions of fluid and solid part of the computational domain. For \( \Gamma(x) = 1 \) discrete forms of Eq. (2) and (3) with the time step \( \Delta t \) reduce to \( \mathbf{u}^* \approx \Delta t \mathbf{u}_s / (\eta + \Delta t) \) and \( T^* \approx \Delta t T_s / (\eta + \Delta t^\eta) \) which for \( \eta \ll \Delta t \) leads to \( \mathbf{u}^* \approx \mathbf{u}_s \) and \( T^* \approx T_s \). Thus, the forcing terms enforce the no-slip boundary conditions and set the required temperature of the solid objects. Simplicity of the IB-VP method has, however, direct consequences in lower solution accuracy. Similarly, as in the classical IB method with a stepwise approach (i.e. without the interpolation [9]) the formal order of the IB-VP method is at most equal to one [8, 10, 11].

Solution algorithm. The solution algorithm for Eqs. (1-3) is formulated in the framework of a projection method [12] for pressure-velocity coupling. The time integration is based on a predictor-corrector approach (Adams-Bashforth/Adams Moulton) and the spatial discretisation is performed using 6th/5th order compact difference and WENO (Weighted Essentially Non-Oscilatory) schemes on half-staggered meshes [13, 14].
3. Experimental facility
The experimental research were performed using a climatic chamber (LTCL600 Series 3, TASLtd., WestSussex, Great Britain), which allowed setting the outside temperature and its exact control during the whole experiment. A sample layer consisted of granular material (expanded clay) filled with air is shown in Fig. 1(a). Its effective heat diffusivity was equal to $\alpha_g = 8.0 \times 10^{-7} m^2/s$. The length ($L_x$), height ($L_y$) and width ($L_z$) of the layer were equal to 0.12m, 0.3m, 0.3m, respectively, and it was insulated from the top and bottom ($y = \pm Ly/2$) and at $z = \pm Lz/2$. The temperature measurements were carried out using thermocouples placed between the granules. They were mounted on specially constructed racks allowing their positioning inside the layer at the selected locations, as shown schematically in Fig. 1(b). The goal of the experimental research was to provide the temperature distribution inside the layers, which could be than used to verify the proposed numerical model.

![Figure 1. Research test stand: a) climatic chamber with a layer of granular material in a close-up, b) the locations of thermocouples in the layer](image)

4. Results
4.1. Assessment of the IB-VP accuracy
This section shows how the theoretical limitation of the solution accuracy of IB-VP method translates on the results obtained for the cases including the heat and mass transfer in the flows in channels and in between spheres. The test cases included: (1) an isothermal flow in a channel filled with six layers of spheres and (2) a flow through a layer $10 \times 10 \times 10$ spheres.

In first test case (1) we considered the flow through the rectangular channel in which six layers of spheres ($D = 28mm$) were placed in the corners, as shown in Fig. 2. The size of the domain was $1D \times 1D \times 15D$. The simulations were performed for Reynolds numbers $Re = UD/\nu = 28.88$ ($U$ - inlet velocity, $\nu$ - viscosity), $Re = 59.78$, $Re = 105.57$ and $Re = 204.74$. The numerical results were compared with experimental data [15] and with the results obtained using the ANSYS Fluent code with 2nd order discretisation. An important factor affecting the solution accuracy is the density of the mesh. To analyse this issue for the present test case the computations were performed for the meshes with significantly different number of nodes. For the SAILOR code the meshes were uniform and consisted of $40 \times 160 \times 40$ (mesh M1), $80 \times 160 \times 80$ (M2) and $80 \times 320 \times 80$ (M2) nodes. In the simulations performed using the ANSYS Fluent code the body-fitted meshes precisely matched the shapes of the spheres and they consisted of approximately $2.5 \cdot 10^6$ (mesh M1) and $7 \cdot 10^6$ (M2) tetrahedral cells. Preparation of the meshes in the ANSYS software required cutting of small cylinders in locations in which the spheres were in contact (see Fig. 2), otherwise the mesh generation resulted in error. Comparisons of the streamwise velocity...
profiles along the $x$-direction from the location in between 5th and 6th sphere are shown in Fig. 2.

The obtained results agree with the experimental data relatively well and the accuracy increases along with the Reynolds number. It seems that the numerical solutions slightly underpredict the measured values in the locations $x/D > 0.25$ for the case with $Re = 28$ and at $x/D = 0.0$ for $Re = 59$. For the higher Reynolds numbers the agreement is very good. It should be noted that the results obtained from the SAILOR and ANSYS Fluent codes practically overlap and are virtually independent of the grid density, which means that the accuracy of the IB-VP method is very good. It seems that lowering the formal order accuracy of IB-VP method to 1st order has no such negative consequences as one could initially presume.

The configuration of the second test case was a simplified model of the granular layer. The computational domain was a rectangular box $0.1 \times 0.3 \times 0.1 m^3$ in which ten layers of $10 \times 10$ hot spheres ($T_s = 400K$, $D = 0.008m$) were placed in the central part of the domain starting from $y = 0.05m$, as shown in Fig. 3. The gaps in between particular spheres were equal to 0.0002m. At the lower boundary we assume uniform velocity $0.1959m$ and temperature $300K$ ($Re = UD/\nu = 100$). The computations were performed with the use of a uniform mesh consisting of $181 \times 240 \times 181$ for the SAILOR code and body-fitted mesh with approximately $11 \times 10^6$ cells for the ANSYS Fluent code. The comparison of the results is presented in Fig. 3(right figures) showing the velocity and temperature profiles at the locations at $y = 0.06m$ (the second row of the spheres) and $y = 0.15m$ (above the layer) along the $x$-coordinate on the lines placed in between the spheres. It can be seen that discrepancies between the results obtained with the help of the SAILOR and ANSYS Fluent codes are small and only quantitative. The locations of maxima/minima of the velocity and temperature are predicted very well and only its absolute values are slightly under-predicted or over-predicted, depending on the localisation and the compared quantity.

### 4.2 Heat and mass transfer in the granular layer

It has been shown that IB-VP approach can be applied for modelling of the heat and fluid flow in complex domains with good accuracy. In this subsection we analyse the heat transfer inside the granular material layer studied experimentally. In the simulations the model of the layer was taken smaller in the vertical ($y$) and horizontal ($z$) direction, i.e., $L_y = 0.04m$, $L_z = 0.04m$, and the domain boundaries at $y = \pm L_y/2$ and at $z = \pm L_z/2$ were assumed to be adiabatic.
Figure 3. Left figure: the computational domain with the contours of temperature in the central cross-section. Right figures: temperature and velocity profiles at $y = 0.06\text{m}$ and $y = 0.15\text{m}$

Taking the smaller layer dimensions for the simulations was necessary because of very high computational costs that would be required for the full scale model. The mesh was uniform and consisted of $256 \times 208 \times 84$ nodes in the $x$, $y$ and $z$ direction, respectively. The preliminary tests have shown that such a mesh was dense enough to obtain qualitatively consistent solutions.

We considered three cases: (1) substitute layer treated as uniform solid material (no flow) with the effective heat diffusivity $\alpha_g = 8.0 \times 10^{-7}\text{m}^2/\text{s}$; (2) non-uniform layer (no flow) represented by spherical $13 \times 5 \times 5$ orderly packed spheres ($D = 10\text{mm}$, $\alpha_g = 5.3 \times 10^{-7}\text{m}^2/\text{s}$) with the empty spaces filled with air ($\alpha_{\text{air}} = 1.9 \times 10^{-5}\text{m}^2/\text{s}$); (3) as the configuration 2) but with the air injected from the lower side of the layer.

Configuration 1. This was the configuration studied experimentally. It provided the temperature distribution inside the layer, which allowed for an additional verification of the applied numerical method. The test was performed as follow. Initially, the temperature of the layer was uniform and equal to $298\text{K}$. Then, the outside temperature on the right side of the layer (see Fig. 1(b)) started to decrease linearly ($T(t) = 298 - 2.083 \times 10^{-3}t \text{[K]}$) such that at the time $16100\text{s}$ it fallen down to $265\text{K}$. Figure 4(a) presents a comparison of numerical results obtained from the SAILOR code and with experimental data as well as the simulation results obtained using the ANSYS Fluent code. It can be seen that the convergence of the numerical solutions is very good. Only slight differences between these results and experimental data prove high precision of the applied codes and solutions algorithms.

Configuration 2. This test case is very similar to the previous one and the only difference is that the granules and air are treated separately. As can be seen in Fig. 4(a) in this case, the differences between the results obtained for configuration 1 are large and increase across the
layer in the $x$-direction. Due to larger heat diffusivity of the air the changes in the temperature distribution occur faster. At the end of the layer at position $x = 0.04m$ at time 12000 the temperature is about 10K lower compared to the case with the effective diffusivity. It seems that in this particular case modelling of the heat flow trough the layer assuming it as homogenous is more accurate than treating it as compound of solid spheres and empty spaces. It can be caused by incorrect specification of the solid material properties provided by the producer.

Figure 4. Comparison of the numerical and experimental results for the granular layer

The complex structure of the layer and thus the problems with the exact placement of measuring devices may be the reason for the slight deviations in the experimental results. Additionally, the case in which the layer was considered as a system composed of solid granulate with the air filling the intergranular spaces (dashed line designated as "gr+air") was analyzed.

Configuration 3. For this configuration two buffer layers were added to the domain, i.e., 0.02m from the bottom and 0.04m from the top, as shown in Fig. 5. They were required to specify the inlet and outlet boundary conditions and to minimise their impact on the flow in between the spheres. As the initial solution we took the results at the time instant 12000s presented in Fig. 4(a). The air with the temperature 300K was injected from the bottom side of the domain with a uniform velocity equal to 0.1m/s. It was higher than the temperature of the layer and this implied that the spheres were heated. Figure 5 on the left side shows the temperature iso-surface inside the layers and vertical velocity contours and velocity vectors in the main cross-section plane for the time moment 4 seconds from the beginning of the air flow. Figure 5(a,b) show the time evolutions of the temperature and velocity fields in the same plane. The words "SPHERE" in the zoomed figure showing the velocity vectors point the locations of the spheres which were removed for a better visibility of the results. It can be seen that the flow structures and temperature distributions are very complex and non-uniform in space. There are recirculation regions (negative velocity), which occur both in between the spheres as well as above the layer. Such an inhomogeneity of the flow around individual spheres enhance the heat exchange process. As one could expect, because of the smaller heat diffusivity of the spheres, they are heated slower than the spaces in between them. Figure 4(b) shows the temperature variations across the layer in two different vertical locations and at two time instances. A wavy character of the profiles is readily apparent and the maxima correspond to the places in between the spheres.

Figure 6 presents the velocity contours in $x-z$ cross-sections planes and $y-z$ cross-sections at $x = 0.065$ (across the spheres) and $x = 0.06$ (in between). It can be seen that the maxima
Figure 5. Left figures show the solution at 4.0s from the beginning of the air flow: temperature iso-surfaces (upper figure) and velocity contours with vectors (lower figure, zoom). Figures on the right side show the temperature and velocity contours at the selected time instances.

Figure 6. Velocity contours in $x-z$ cross-sections (left figures) and $y-z$ cross-sections at $x = 0.065$ (across the spheres) and $x = 0.06$ (in between).

of the velocity occur in the centres of the regions in between the spheres. In these locations the flow strongly accelerates and compared to the inlet conditions the velocity increases almost
10 times. The streams of the air with high velocity extend even above the layers, as can be seen in Fig. 6(right figures). The complex structure of the flow behind the spheres (plane at $x = 0.065$ - across the spheres) manifests by recirculation regions denoted by white colour. It can be seen that these regions appear already behind the first row and their shapes do not change significantly across the layer. This means that the flow structure inside the layer at different vertical locations is similar, and hence, its impact on the heat transfer process on particular row of the sphere is also similar.

5. Summary
This paper presented numerical and experimental studies on the heat and flow in granular layers. The IB-VP method applied in the research was proven to be sufficiently accurate to predict the inhomogeneities of the granulates and complex mass and heat transfer phenomena in between the inter-granular spaces. This was demonstrated by comparisons with the experimental data and the numerical data obtained using the ANSYS Fluent code. The results obtained for the granular layers revealed existence of various size of recirculation regions developing behind the granulates (spheres). Their presence intensifies the heat exchange process between the solid and gas phases. It was shown that the temperature inside the layers can be effectively changed and controlled by the parameters of air flowing through them. Note that such air can be taken from the air-conditioning system, for instance.

Acknowledgements
This work was supported by National Science Centre, Poland (Grant no. 2017/27/N/ST8/02318) and statutory funds of Czestochowa University of Technology under BS/MN 1-103-301/2018/P. PL-Grid infrastructure was used to carry out the computations.

References
[1] Vargas W L and McCarthy J J 2001 Heat Conduction in Granular Materials AIChE J 47 1052-1059
[2] Tian K S and Shu H J 2009 Progress in porous media research Nova Science Publishers Inc. New York
[3] Amhalhel G A and Furmanski P 1997 Problems of modeling flow and heat transfer in porous media Buletyn Instytutu Techniki Cieplnej Politechniki Warszawskiej 85
[4] Sosnowski M, Krzywanski J, Grabowska K and Gnatowska R 2017 Polyhedral meshing in numerical analysis of conjugate heat transfer Proceedings of International Conference on Experimental Fluid Mechanics (EFM) 589-94
[5] Breugem W P, van Dijk V and Delfos R 2014 Flows Through Real Porous Media: X-Ray Computed Tomography, Experiments, and Numerical Simulations J. Fluids Eng. 136(4) 040902
[6] Szymianek E, Blaszczyk T, Hall M R, Dehdezi P K and Leszczynski J 2014 Modelling and analysis of heat transfer trough 1D complex granular system Granular Matter 16(5) 687-694
[7] Mittal R and Iaccarino G 2005 Immersed boundary methods Ann. Rev. Fluid Mech. 37 239-261
[8] Khadra K, Angot P, Parneix S and Caltagirone J P 2000 Fictitious domain approach for numerical modelling of Navier-Stokes equations Int. J. Numer. Meth. Fluids 34 651-684
[9] Fadlun E A, Verzicco R, Orlandi P and Mohd-Yusof J 2000 Combined Immersed-Boundary Finite-Difference Methods for Three-Dimensional Complex Flow Simulations J. Comp. Phys. 161 35-60
[10] Kadoch B, Kolomenskiy D, Angot P and Schneider K 2012 A volume penalization method for incompressible flows and scalar advection diffusion with moving obstacles J. Comp. Phys. 231 4365-4383
[11] Tyliszczak A and Ksiezyk M 2018 Large eddy simulations of wall-bounded flows using a simplified immersed boundary method and high-order compact schemes Int. J. Num. Meth. Fluids 87 358-381
[12] Fletcher C A J 1991 Computational Techniques for Fluid Dynamics Springer-Verlag
[13] Tyliszczak A 2014 A high-order compact difference algorithm for half-staggered grids for laminar and turbulent incompressible flows J. Comp. Phys. 276 438-467
[14] Tyliszczak A 2016 High-order compact difference algorithm on half-staggered meshes for low Mach number flows Comput. Fluids 127 131-145
[15] Suekane T, Yokouchi Y and Hirai S 2003 Inertial flow structures in a simple-packed bed of spheres AIChE J. 49 10-17.