Immersed boundary method (IBM) based direct numerical simulation of open-cell solid foams

Citation for published version (APA):

DOI:
10.1002/aic.15487

Document status and date:
Published: 01/03/2017

Document Version:
Publisher’s PDF, also known as Version of Record (includes final page, issue and volume numbers)

Please check the document version of this publication:
• A submitted manuscript is the version of the article upon submission and before peer-review. There can be important differences between the submitted version and the official published version of record. People interested in the research are advised to contact the author for the final version of the publication, or visit the DOI to the publisher's website.
• The final author version and the galley proof are versions of the publication after peer review.
• The final published version features the final layout of the paper including the volume, issue and page numbers.

Link to publication

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain
• You may freely distribute the URL identifying the publication in the public portal.

If the publication is distributed under the terms of Article 25fa of the Dutch Copyright Act, indicated by the “Taverne” license above, please follow below link for the End User Agreement:
www.tue.nl/taverne

Take down policy
If you believe that this document breaches copyright please contact us at:
openaccess@tue.nl
providing details and we will investigate your claim.

Download date: 15. Sep. 2020
Immersed Boundary Method (IBM) Based Direct Numerical Simulation of Open-Cell Solid Foams: Hydrodynamics

Saurish Das
Dept. of Chemical Engineering and Chemistry, Multiphase Reactors Group, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

Niels G. Deen
Dept. of Mechanical Engineering, Multiphase and Reactive Flows Group, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

J. A. M. Kuipers
Dept. of Chemical Engineering and Chemistry, Multiphase Reactors Group, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

DOI 10.1002/aic.15487
Published online September 16, 2016 in Wiley Online Library (wileyonlinelibrary.com)

A sharp interface implicit immersed boundary method is developed and used for direct numerical simulations of the flow through open-cell solid foams with a cellular structure. The complex solid structure of the foam is resolved on a non-boundary fitted Cartesian computational-grid. A single representative unit cell of the foam is considered in a periodic domain, and its geometry is approximated based on the structural packing of a tetrakaidecahedron. Simulations are performed for a wide range of porosities (0.638–0.962) and Reynolds numbers (0–500). Flow is enforced by applying a constant body force (momentum source) for three different flow directions along the \{100\}, \{110\}, and \{111\} lattice-vectors. The drag force on the foam is calculated and a non-dimensional drag/pressure drop correlation is proposed that fits the entire data set with an average deviation of 5.6%. Moreover, the accurate numerical simulations have helped to elucidate the detailed fluid-solid interaction in complex porous media.

Keywords: open-cell solid foam, porous media, direct numerical simulation, immersed boundary method, drag correlation

Introduction

There has been an increasing trends on the use of novel materials to improve the process efficiency in a cost-effective way and to minimize the total weight/volume of process equipment. Open-cell solid foams is one of such material which is extensively used over the past few decades to form porous media for its excellent thermal, mechanical, and acoustics properties. These foams are cellular solid structures made of metal (e.g., aluminum, nickel, iron-chromium, etc.) or ceramics (e.g., \text{Al}_2\text{O}_3, silicon, carbide, etc.) Due to its large surface area to volume ratio it has ample applications in heat-transfer devices like heat exchangers, thermal energy absorbers, vaporizers, heat-shielding devices, etc. Moreover, foams are also gaining popularity in several other niche applications like high-temperature filters, pneumatic silencers, burn rate enhancers etc. In fixed-bed catalytic reactors open-cell foams are gaining attention as catalyst support which improves gas-liquid contacting to enhance the heat and/or mass-transfer rate with minimal pressure drop as compared to other conventional packings. In addition, in cases of exothermic reaction systems where heat needs to be removed from the bed, the use of metallic foams enhances the heat removal rate. Based on the type of application and the size of the reactor solid foams are used as a whole block to fill the entire bed or it is applied in the form of pellets that are dumped in the reactor.\(^1\) Designing, optimizing, and finding the favorable operation conditions of such processes can be achieved with the help of more coarse-grained macroscopic numerical models. However, these coarse-grained models require a quantitative closure that captures the underlying detailed hydrodynamic behavior at the pore-scale level. Such a closure describes the momentum exchange between the solid foam and the percolating fluid, i.e., a drag or pressure drop correlation, and can be obtained from detailed pore-scale level studies as presented in this work.

Due to the complex and random geometrical shapes of solid foams, most of the reported work in literature dealing with solid foams is experimental. A large number of experimental studies were reviewed by Edouard et al.\(^2\) and Dietrich.\(^3\) They reported very large deviations between the existing pressure drop correlations due to (1) morphological differences between the different commercially available solid foams, and (2) experimental

Correspondence concerning this article should be addressed to Niels G. Deen at N.G.Deen@tue.nl.

This is an open access article under the terms of the Creative Commons Attribution-NonCommercial-NoDerivs License, which permits use and distribution in any medium, provided the original work is properly cited, the use is non-commercial and no modifications or adaptations are made.

© 2016 The Authors AIChE Journal published by Wiley Periodicals, Inc. on behalf of American Institute of Chemical Engineers
inaccuracies. At the macroscopic level there also exist a few analytical and theoretical studies where very simplified porous geometries with volume-averaged properties are used to study open-cell foams.\(^1\)\(^-\)\(^4\) In contrast, numerical studies at pore-scale level are capable to overcome all experimental uncertainties and to precisely account for specific geometrical details. At the same time arbitrary material properties and flow conditions can be defined and perfectly controlled in simulations. Many researchers investigated open-cell foams numerically by resolving the detailed solid structure in a 3D computational domain, but the studied parameter spaces does not cover the entire operational range of catalytic packed bed reactors. Mainly, previous studies concentrated on foams of high porosity, since most of the commercially available foams are highly porous. However, a catalyst-washcoat layer decreases the porosity of the foams. Both Chung et al.\(^10\) and Horneber et al.\(^11\) used a tetrakaidecahedron geometry (Kelvin’s unit cell) with a cylindrical strut to represent the foam unit cell. They performed simulations only for \(\varepsilon=0.95\) and \(\varepsilon=0.89\), respectively. Lucchi et al.\(^12\) used both idealized and randomized Kelvin’s unit cell geometries and studied porous system of \(\varepsilon=0.80, 0.84, 0.87,\) and 0.90. The Weaire-Phelan cell has been used by Boomsma et al.\(^13\) to represent foam unit cell of \(\varepsilon=0.96\). Bai et al.\(^14\) created a foam unit cell by geometrically extracting a solid sphere from a solid tetrakaidecahedron structure. They presented simulation results for \(\varepsilon=0.97\). Solid structures formed by the void space between BCC and FCC arrangements of spheres, and also an A15 lattice in a cube has been used as foam structure by Krishnan et al.\(^15\) In their studies the porosity was ranging from 0.8 to 0.98. Similar unit cell shape have been used by Suleiman et al.\(^17\) to perform simulations on large blocks of foams of \(\varepsilon=0.89, 0.91,\) and 0.97. In all of these studies, unstructured grid based CFD codes have been used. The unit cell is placed in a channel or a periodic box, only a single flow direction was considered. However, in real situations the orientation of the unit cell will usually be random and the flow direction will affect the overall drag force. We also notice that most pressure drop correlations were presented in an Ergun type form as \(\Delta P/L=au_r+bu_r^2\), where \(a\) and \(b\) only depend on \(\varepsilon\). Only in a few studies\(^18\)\(^-\)\(^20\) porous media (including our present simulation results) is suggested that \(b\) is only independent of \(Re\) at sufficiently high values of \(Re\). As a result, these correlations yield a large error at low \(Re\).

One step ahead from idealized unit-cell assumption, Kopanidis et al.\(^21\) and Suleiman et al.\(^22\) have used Surface Evolver software\(^23\) that follows the actual foam forming process (i.e., minimization of surface energy) to generate their foam geometry. Recently, several authors\(^24\)\(^-\)\(^29\) also presented accurate, assumption free numerical results for real foam structures where a 3D foam image captured from a micro-CT scan has been converted into a CFD-readable geometry. However, those results are only valid for a particular geometry. There is no generalized drag correlation available in the literature that is valid for a wide range of parametric values, even not for idealized foam geometries. Hence, the main objective of this contribution is to numerically study the flow behavior through solid foams at the pore scale level considering an idealized geometry using a wide range of parameter values.

The outcomes of the present contribution are (1) an efficient, accurate, and robust immersed boundary method (IBM) based direct numerical simulation framework to fully resolve the flow at the pore scale level of the foam structure, (2) a detailed understanding of hydrodynamics through such complex porous media, and (3) a pressure drop or drag correlation for a wide range of porosities and Reynolds numbers, which can be used in more coarse-grained simulations of large scale processes.

In this work the foam geometry is approximated based on the structural packing of a tetrakaidecahedron (Kelvin’s unit cell\(^30\)) with cylindrical strut morphology. Lord Kelvin proposed that the minimum foam surface for a given volume has a tetrakaidecahedron shape. This shape is commonly called Kelvin’s unit cell, and this shape is most likely attained in the production process of solid foams.\(^31\) To resolve this complex unit-cell geometry in a 3D Cartesian periodic computational domain, a sharp interface based direct/implicit IBM, originally proposed by Deen et al.\(^32\) has been adopted and further extended. The applied IBM is a novel computational technique that is a potential alternative for body conformal grid methods. It avoids the time-consuming high-quality grid generation process and implementation of the Navier Stokes (N-S) solver on complex unstructured grids. Simple structured grid based techniques offer considerable advantages over unstructured grid in terms of efforts of code development, memory requirement to store the grid information, computational efficiency and the manpower involved to generate good quality meshes. Conversely, body fitted grids have a great ability to locally refine the mesh in zones of interest where sharp gradient occur. There exists a large variety of numerical schemes to solve the Navier-Stokes equations in non-body-conformal grids, such as the overset grid method\(^33\) the Lattice Boltzmann method (LBM),\(^34\)\(^35\) Lagrange-multiplier-based fictitious-domain method (DLM),\(^36\)\(^37\) force-coupling method,\(^38\) several particle based (Lagrangian) methods like Smooth Particle Hydrodynamics or Moving Particle Semi-implicit method.\(^39\) There exists a large variety of IBM implementations and an excellent literature review was reported by Mittal and Iaccarino.\(^40\) The existing methods can be classified into mainly two categories, namely momentum forcing approach and ghost cell approach. In the momentum forcing approach, the no-slip condition at the immersed boundary (IB) was applied by introducing a forcing function as a source term in the momentum equation. The value of this forcing function is maximum at the location of the solid surface and distributed only to the vicinity of the immersed boundary (by means of a smoothed Dirac Delta Function). Calculating the value of the forcing function is the main crux of this method. The momentum forcing can be done in a continuous manner or in a direct manner. The continuous forcing approach was first proposed by Peskin.\(^41\) In this case, the forcing term is calculated in a feedback loop by taking the difference between fluid velocity at the IB surface and the desired velocity at the IBM surface.\(^42\)\(^-\)\(^45\) Conversely, in the direct forcing approach the forcing term is not obtained by any kind of feedback mechanism, rather directly in the momentum balance at the forcing points placed at the IB surface. Different variations of the direct forcing approach can be found in the literature.\(^46\)\(^-\)\(^48\) Both of these momentum forcing formulations require the solution of the governing equations inside the immersed body; and also interpolation between the forcing points and the background grid. On the other side, the ghost-cell method is more elegant where the no-slip condition is forced via ghost cells (cells near to the IB),\(^32\)\(^35\)\(^35\)\(^35\) All boundary conditions on the immersed boundary are enforced at the level of the discretized momentum equation by means of pre-calculated flow variables (based on geometrical interpolations) at the ghost-cells. The flow field does not need to be solved inside the IB and can be used for very complex solid structures. The cut-cell finite volume (FV)\(^49\)\(^-\)\(^50\) approach is an another kind of IBM which generates
body conformal cells around the IB surface, formed by the
decomposition of the near body Cartesian cells cut by the IB
surface. The Navier-Stokes equations are solved for all those
“cut cells” with proper flux calculations through the complex
faces created by the intersection of the solid surface and the
Cartesian control volume (CV). As a result, it offers improved
global and local conservation of mass and momentum as com-
pared to other methods. However, extending this approach in
3D, reshaping of CVs and discretization of the full Navier-
Stokes equations is very difficult. Our present implementation
of direct implicit IBM is motivated by Deen et al.,32 found to
be second order accurate, stable and easy to implement in a
FV code.

This article is organized as follows. First, we describe the
numerical model and implementation of the current IBM. Sub-
sequently, a detailed validation of the current numerical model
is reported. After introducing the foam geometry and its main
morphological parameters, the simulation results are pre-
tented. Finally, a drag or pressure drop correlation for open-
cell solid foams is presented in terms of non-dimensional
quantities and compared with available experimental results.

Numerical Details
Numerical model and solution methodology
The conservation equations for mass and momentum are
solved in 3D Cartesian domain. For an incompressible, New-
tonian fluid it reads
\[
\nabla \cdot \bar{u} = 0 \quad (1)
\]
\[
\rho \frac{\partial \bar{u}}{\partial t} + \rho \nabla \cdot (\bar{u} \bar{u}) = -\nabla p + \nabla \cdot \tau \quad (2)
\]
\[
\nabla \bar{u} = (\nabla \bar{u})^T - \frac{2}{3} \mathbf{I} \nabla \cdot \bar{u}
\]
is fluid stress tensor. In the current
FV implementation transport equations are integrated for each
staggered computational CV
\[
\frac{\partial}{\partial t} \left[ \int_V \bar{u} \, dV \right] + \left[ \int_A \rho \bar{u} \, dA \right] = - \left[ \int_A p \, dA \right] + \left[ \int_A \tau \cdot \bar{n} \, dA \right]
\]
(3)
where \( k \) indicates the time level and \( C \) indicates the convective
flux: \( \bar{C} = (\bar{u} \cdot \bar{n}) \bar{u} \). A deferred correction method has been
incorporated in the convection term, where both the First
Order Upwind (FOU) scheme and the total variation diminish-
ing (TVD) Min-mod scheme are used to calculate the convection
flux. The flux based on the FOU scheme \( (C_{FOU}) \) is used as a predictor and treated implicitly, while the difference
between the TVD and FOU schemes is used as corrector that is
treated in an explicit manner. Second order accurate Central
Difference scheme is used for discretizing the diffusion terms.
The Eq. 4 is solved in staggered computational grid by a frac-
tional step method, where initially the tentative velocity field
\( \bar{u}^{*+} \) is computed using old time level pressure value \( (p^*) \) from
\[
\rho \bar{u}^{*+} = \rho \bar{u}^k + \frac{\Delta t}{\Delta V} \left\{ - \int_A p^{*+} \bar{n} \, dA - \rho \left[ C_{FOU}^{k+1} + \left( C_{Min-Mod} - C_{FOU}^{k+1} \right) \right] dA \right\}
\]
(5)
Note that The stress tensor is split into an implicit part and an
explicit part. In the momentum equation for the \( x \) direction,
only the \( x \) velocity component is treated implicitly, whereas the
other velocity components are treated explicitly. As a result Eq.
5 forms three decoupled sets of linear equations. To find \( \bar{u}^{*+} \)
we use a robust and efficient parallel Block-Incomplete Cholesky
Conjugate Gradient (B-ICCG) solver. The velocity at the new
time level \( k + 1 \) can subsequently be obtained from
\[
\bar{u}^{k+1} = \bar{u}^{*+} - \frac{\Delta t}{\rho} \nabla (\delta p)
\]
(6)
which is again solved by the B-ICCG sparse matrix solver.
For 3D non-periodic cases the sparse matrices for velocity and
pressure (Eqs. 5 and 7) contain one central and six neighboring
bands in the sparse matrix (for 3D flow, periodic in all three
directions) it provides a numerically stable and accurate
solution.
Immersed boundary method

In this section, we explain the implementation of the present IBM for the case of 3D spherocylindrical shaped objects. The applied IBM is based on the method described by Deen et al. and schematically shown in Figure 1. A total of 36 spherocylinders are used to prepare one single unit-cell in a 3-D periodic domain (Figure 11). To enforce the no-slip boundary condition at the impermeable wall, at first, all the cells are marked/flagged as either “solid-cell” (cell-center inside the solid body) or “fluid-cell” (cell-center in the fluid zone) in a staggered computational grid. “IB-cells” are a special type of fluid cells, as they have at least one neighboring solid-cell. It is in these IB-cells that the boundary condition is imposed. The algebraic discretized form of momentum equation (Eq. 5) at cell-centers $c$ can be represented as

$$a_c \phi_c + \sum_{nb} a_{nb} \phi_{nb} = b_c$$  \hspace{1cm} (12)

where $\phi$ corresponds to one of the fluid velocity components and $nb$ indicates surrounding neighboring cells. $a$ represents the fluid property and geometry dependent coefficients and all the explicit terms are contained in $b$. If a cell $c$ is an IB-cell, at least one of the surrounding cells $nb$ is a solid-cell, and the no-slip condition is imposed by changing the central coefficient ($a_c$) and neighboring coefficient($a_{nb}$) of Eq. 12. Figure 1 shows a single spherocylindrical solid body in the Cartesian computational grid where IB-cell $c$ is neighbored by a solid-cell $w$. The value of $\phi$ in the solid-cell can be eliminated from the discretized equation by expressing it as a linear combination of relevant fluid-cells in such a way that it satisfies the no-slip boundary condition given by $\phi_w$. For this purpose a quadratic fit is used: $\phi = p\xi^2 + q\xi + r$, where $\xi$ is the non-dimensional distance from solid-cell $w$, shown in Figure 1. The values of coefficients $p$, $q$, and $r$ are obtained from the known values of $\phi$ at the particle boundary (at $\xi = \xi_1$, $\phi = \phi_1 = 0$) and the two nodes in the fluid (at $\xi = 1$, $\phi = \phi_e$ and $\xi = 2$, $\phi = \phi_e$). $\xi$ corresponds to the distance from the solid-cell $w$ to the point where the grid line intersects the solid surface ($s$). This gives

$$\phi_w = \frac{-2\xi}{1-\xi} \phi_e + \frac{2-\xi}{2-\xi} \phi_e$$  \hspace{1cm} (13)

The modified coefficients ($\tilde{a}$) for Eq. 12 can then be written as

$$\tilde{a}_c = a_c + a_w \left( \frac{-2\xi}{1-\xi} \right)$$  \hspace{1cm} (14)

$$\tilde{a}_e = a_c + a_w \left( \frac{-\xi}{2-\xi} \right)$$  \hspace{1cm} (15)

$$\tilde{a}_w = 0$$  \hspace{1cm} (16)

As a result, all the solid-cells get decoupled from the main flow domain as their coefficients are all zero. Note that Eq. 12 is also formed for solid-cells and to enforce zero velocity the central coefficient ($a_c$) is modified to 1 and all other terms (i.e., $a_{nb}$, $b_c$) are set to zero. The above procedure needs to be carried out for all the three velocity components: but no strict treatment (like $\partial O/\partial s = 0$ at the solid wall) is carried out for the pressure field as the velocity field inside the solid body is already divergence free. The pressure field is also computed inside the solid body and the pressure correction step (Eq. 6) leads to a very small value of the velocity $(O(u \cdot 10^{-3}))$ inside the solid body, which has an insignificant effect on the overall simulation results. In Eq. 14, $\tilde{a}_e$ becomes singular when $\xi_e \to 1$, i.e., in Figure 1, $c$ is at the solid surface or very close to it. For such cases $c$ is assigned as solid-cell and $e$ to IB-cell with $\xi_e = 0$.

It is clear from the above method, proposed by Deen et al., that the main tasks of this IBM implementation are to properly mark the cells (i.e., fluid or solid-cells) and to find the $\xi$ for all IB-cells. Four scalar fields ($f$) are used to mark the CVs for the pressure and the free velocity components, each stored at different (staggered) grid locations. All the fluid-cells are marked with $f = 0$ and solid-cells are marked based on predefined solid body index, such that from the unique cell-flag value ($f$) the program can identify for which solid body to calculate $\xi$ and change the coefficients based on Eqs. 14–16. For a solid spherocylinder with index $p$, the cylindrical part is indicated with $f = p$, whereas the two hemispherical parts are flagged with $f = p + 0.25$ and $f = p + 0.50$. IB-cells are identified during the solution procedure by checking the flags of the six neighbors of individual fluid-cells. In Figure 1, any cell $w$ will be inside the cylinder $\bar{AB}$ when the initial point of the normal from $w$ to the axis of the cylinder $(\bar{AB})$, i.e., $N_w$ lies on the line segment $\bar{AB}$ and the normal distance ($l_w$) is less than radius ($R$) of the cylinder

$$|\bar{AN}_w| + |\bar{BN}_w| = |\bar{AB}| \text{ and } l_w = \frac{|(\bar{X}_A - \bar{X}_B) \times \bar{X}_B - \bar{X}_w|}{|\bar{X}_A - \bar{X}_B|} \leq R$$

where $X$ represents the position vector of the point. Similarly for the hemispherical parts, any cell $p$ will be inside any of the hemispheres (say hemisphere with center $B$) if it satisfies the following conditions

$$|\bar{AN}_p| + |\bar{BN}_p| \neq |\bar{AB}|, \text{ and } |\bar{AN}_p| \geq |\bar{BN}_p|$$

and $|\bar{X}_B - \bar{N}_p| \leq R$

where $N_p$ is the initial point of the normal from $p$ to line $\bar{AB}$, as shown in the figure. Once all the cells are flagged properly
based on solid body index, the next task is to find values of $\xi_c$. For hemispheres it can be found analytically in a straightforward way. However, for the cylindrical part it is calculated iteratively. Say the normal distance from IB-cell $c$ and its neighboring solid-cell $w$ to line $\overline{AB}$ are $l_c$ and $l_w$, respectively. Hence, $l_c \geq R$ and $l_w \leq R$. We need to find the point of intersection between the line segment $\overline{CW}$ and the solid wall. To this end, we use the method of false position or the bisection method to find the unique point on line segment $\overline{CW}$ from which normal distance ($l_c$) to the line $\overline{AB}$ is equal to $R$. It is found that for almost all the IB-cells a maximum number of 3–5 iterations is required to get an accuracy in the order of $0.5 \times 10^{-7}$ in $\Delta x$, where $\Delta x$ is the grid resolution.

It may also be possible for an IB-cell to have multiple neighboring cells that are solid-cells. Figure 2 shows a case of two intersecting spherocylinders, where IB-cell $C_1$ neighbors two solid-cells $e$ and $s$ in the east and south direction. Due to the presence of the solid-cell $e$ in the $x-$ sweep the modified coefficient ($\tilde{a}_{ew}$, $\tilde{a}_{sw}$ and $\tilde{a}_{ew} = 0$) are calculated based on old coefficients; and also for the presence of solid-cell $s$ in the $y-$ sweep modified coefficients ($\tilde{a}_{sw}$, $\tilde{a}_{es}$, and $\tilde{a}_{sw} = 0$) are calculated from $\tilde{a}_{sw}$, $\tilde{a}_{es}$, and $a_{sw}$. It can be shown that in this case the final coefficients ($\tilde{a}$) do not depend on the order in which the coefficients are modified, because the solid-cells and the IB-cell do not lie on a single line. However, for IB-cell $C_2$ two solid-cells $w$ and $e$ do lie on the same grid line and modifications become sequence dependent. To avoid this for such few special IB-cells we use a linear fit ($\phi = q \xi + r$) instead of a quadratic fit and the coefficients are modified as follows

$$\phi_w = \frac{\xi_{sw}}{1 - \xi_{sw}} \phi_{C_2}$$

$$\tilde{a}_{ew} = a_{ew} + \frac{\xi_{sw}}{1 - \xi_{sw}} a_w$$

$$\tilde{a}_{sw} = a_{sw} + \frac{\xi_{sw}}{1 - \xi_{sw}} a_{ew}$$

$$\tilde{a}_{sw} = 0$$

$$\tilde{a}_{es} = 0$$

where $\xi_{sw}$ and $\xi_{ew}$ correspond to the non-dimensional distance between solid wall and solid-cells $w$ and $e$, as shown in Figure 2. Since there are only very few such cells, the simulation still retains second order accuracy. Note that if an unstructured mesh would have been used to describe nontrivial details of the geometry (small gaps/sharp angles) this would lead to highly skewed unstructured cells, creating difficulties in convergence. The present methodology is free from such issues.

**Verification**

The current implementation is thoroughly tested and verified with existing numerical and/or experimental results along with established empirical correlations. We have first considered flow past a cylinder in an infinite domain. The drag force, unsteady vortex shedding frequency and lift force are calculated and compared with literature data. After that, flow past a sphere (3D case) is simulated for different Reynolds number and drag force exerted on the sphere is compared. Flow through simple cubic (SC) arrays of spheres are simulated for the system of different porosity and Reynolds number.

**Flow past a circular cylinder**

The flow around a circular cylinder has become the standard for testing the accuracy of many computational techniques due to the wide variety of available experimental and numerical results. In this case, the flow field remains steady and symmetric up to $Re = 47$, and for moderate and higher $Re$ it becomes dynamic, leading to periodic von-Karman vortex shedding. The flow field can be assumed to be approximately 2D up to $Re = 200$ (more precisely $Re = 180$). Beyond that the flow becomes 3D and deviations obtained for a 2D solution become significant. We have performed pseudo-2D simulation for $Re = 10, 20, 30, 40, 50, 60, 100, 150, 200$, and 200 where the flow field can still be assumed to behave 2D. The present results are compared with existing body-fitted grid based numerical results, IBM based numerical results, and experimental results. The drag coefficient ($C_D$), lift coefficient ($C_L$), and for the high $Re$ unsteady cases also the Strouhal number ($St$) are compared. These are defined as

$$C_D = \frac{F_D}{(1/2) \rho U^2 \pi D}$$

$$C_L = \frac{F_L}{(1/2) \rho U^2 \pi D}$$

$$St = \frac{fD}{U \pi}$$

where $F_D$ and $F_L$ are the drag force and lift force per unit length of the cylinder; $\rho$, $D$, and $U_\infty$ are the density of the fluid, diameter of the cylinder, and the uniform free stream velocity, respectively. $f$ is the frequency of vortex shedding, obtained by Fast Fourier Transforming the time dependent lift force, $A_p$ is the projected area, equal to $D$ for the case of the cylinder and equal to $\pi D^2/4$ for the case of the sphere. $F_D$ and $F_L$ are calculated by surface integration of the shear stress and pressure force at the cylinder wall as detailed in Deen et al. A pseudo-2D rectangular computational domain is considered, where only three grid layers are used in the direction of the cylinder axis. The domain size in the flow direction is taken as $30D$ and in the other direction as $15D$. The center of the cylinder is placed at a distance of 16.5D from the inlet and at in the
middle between the other two boundaries. A uniform plug flow velocity of $U_\infty$ is specified at the inlet, a prescribed pressure boundary condition at the outlet and at all other boundaries free-slip boundary conditions are used. To check the grid convergence, four different grid resolutions: G10 (10 grid points across the diameter of the cylinder), G20 (20 grid points), G40 (40 grid points), and G60 (60 grid points) have been used for $Re = 40$ (steady state). For $Re = 200$, where flow filed is unsteady, simulation needs to run for very long period, and to be restricted in a feasible computational time simulation is performed for only G20, G30, and G40 grid resolutions. It is found that G20 provides grid independent results (the total deviation is less than 1.5% compared to finest grid, cf. Table 2) and hence the rest of the simulations are performed with the G20 grid. The detailed simulation settings are listed in Table 1.

In Figure 3 the $x-$ velocity field for varying $Re$ numbers is shown. For this current set of Reynolds numbers unsteady behavior is first observed at $Re = 50$ and it becomes pronounced at $Re = 100$ and $Re = 200$, as seen from Figure 3. In Figure 4 and in Table 2 non-dimensional drag coefficient (time averaged for high $Re$ cases) is compared with existing literature. For $Re = 100$ and $Re = 200$, in Table 3 the magnitude of fluctuating lift coefficient, rms-averaged lift coefficient, and also Strouhal number ($St$) of vortex shedding are compared with existing numerical and experimental results. Figure 5 shows a very good agreement of the Strouhal number ($St$) of vortex shedding with other published results.

Over a range of $Re$, a very good agreement is found with both existing numerical and experimental results. At low $Re$, the comparatively large part of the simulation domain gets affected by the presence of the stationary object. This demands a large computational domain to get a domain size independent result. In the present case at low $Re$ the deviation is rather high due to the use of the different computational domain size.

In Figure 6 shows the streamline pattern of flow for $Re = 40$. The symmetric recirculation zone is clearly visible and length of the recirculation zone $L_w$ is defined as the distance between the two stagnation point downstream of the cylinder. To check the accuracy of the local flow features obtained from the current method, the non-dimensional length of the recirculation zone, i.e., $L_w/D$ has been compared with other numerical results in Table 4. Despite large deviation in existing literature results, a reasonable agreement has been observed.

Moreover, to check the effect of time-step on steady-state solution, for $Re = 40$, simulation has been performed using CFL = 0.025 and CFL = 0.25. A very negligible difference of 0.028% has been obtained on drag coefficient. Hence, it can be assured that the current results are independent of time-step size.

### Flow past a sphere

Another classical benchmark problem is the flow past a static sphere placed in an infinite domain. The flow field remains steady up-to $Re = 210$, gets unsteady and symmetric for $Re=210–280$, and beyond that the flow becomes unsteady and unsymmetrical. In our case Reynolds number has been varied from 10 to 500. Simulation snapshots are shown in Figure 7. A 3D domain of $20D \times 20D \times 20D$ for $Re=10$, 20, 30, 40, 50, and 100 is considered and the center of the sphere is placed at the location of $[8D, 10D, 10D]$. The lower Reynolds number flows require comparatively larger domain size and the current domain size is sufficiently high. For higher Reynolds number cases, the simulations need to perform

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values</th>
<th>units</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid resolution</td>
<td>G10 : $300 \times 150 \times 3$</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>G20 : $600 \times 300 \times 3$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G40 : $1200 \times 600 \times 3$</td>
<td></td>
</tr>
<tr>
<td></td>
<td>G60 : $1800 \times 900 \times 3$</td>
<td></td>
</tr>
<tr>
<td>$\Delta x = \Delta y = \Delta z$</td>
<td>G10 : $1 \times 10^{-3}$, G20 : $5 \times 10^{-3}$, G40 : $2.5 \times 10^{-3}$, G60 : $1.667 \times 10^{-3}$</td>
<td>m</td>
</tr>
<tr>
<td>Diameter (D)</td>
<td>0.1</td>
<td>m</td>
</tr>
<tr>
<td>Viscosity</td>
<td>0.1</td>
<td>Pa s</td>
</tr>
<tr>
<td>Density</td>
<td>10, 20, 30, 40, 50, 60, 100, 150, 200</td>
<td>kg/m$^3$</td>
</tr>
<tr>
<td>$U_\infty$</td>
<td>1</td>
<td>m/s</td>
</tr>
<tr>
<td>Time step</td>
<td>$2.0 \times 10^{-7}$</td>
<td>s</td>
</tr>
</tbody>
</table>

**Table 1. Simulation Settings for Flow Past a Circular Cylinder**

**Table 2. Comparison of the Present Computation of Drag Coefficient ($C_D$) with Other Numerical and Experimental Results for Flow Past a Cylinder**

<table>
<thead>
<tr>
<th>$Re$ (Present)</th>
<th>$C_D$ (Experimental)</th>
<th>$St$</th>
<th>$L_w/D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>2.960</td>
<td>1.760</td>
<td>1.601</td>
</tr>
<tr>
<td>20</td>
<td>2.105</td>
<td>1.60</td>
<td>1.562</td>
</tr>
<tr>
<td>50</td>
<td>1.428</td>
<td>1.37</td>
<td>1.54</td>
</tr>
<tr>
<td>60</td>
<td>1.340</td>
<td>1.31</td>
<td>1.46</td>
</tr>
<tr>
<td>100</td>
<td>1.359</td>
<td>–</td>
<td>1.39</td>
</tr>
<tr>
<td>150</td>
<td>1.340</td>
<td>–</td>
<td>1.37</td>
</tr>
<tr>
<td>200</td>
<td>G10 : 1.390</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>G20 : 1.361</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>G40 : 1.346</td>
<td>–</td>
<td>–</td>
</tr>
</tbody>
</table>

Unless specified otherwise all reported data are obtained with the G20 grid.
comparatively very large domain and size of the domain has been restricted to reduce computational burden without losing accuracy of the result. For Re = 100 simulation also has been performed in a comparatively smaller computational domain of 10D (flow direction)×8D×8D and the center of the sphere is placed at the location of [3.75D, 4D, 4D]. The calculated drag force for these two simulations (at Re = 100 in two different domain) only differ by 1.9% (cf. Table 6) and it is expected that with increasing Reynolds number this difference will reduce. Hence, it can be concluded that for Re = 300, 350, and 500 domain size of 10D×8D×8D is reasonable. The simulation settings are listed in Table 5. Here too, four different grid resolutions were used: G10 (10 grid point across the diameter of the sphere), G20, G40, and G60 for Re = 100; and for Re = 500 only G10, G20; and G30 grids are used. As mentioned earlier, on higher Reynolds number, the simulation needs to run for long to reach quasi-steady state and as a result the we have restricted G60 simulation for Re = 500. It is found that G20 is a sufficient (deviation less than 2.5% compared to finest grid) resolution to obtain grid independent results and hence the rest of the simulations are performed using G20. In Figure 8 and Table 6 the computed drag coefficients are compared with experimental the curve fit correlation by Clift et al. and numerical results of Mittal et al. Our results show an excellent agreement over the entire range of Re. 

**Flow past periodic static arrays of spheres at stokes flow regime**

In the last test case we consider flow through a simple cubic array of spheres. This case is numerically very similar to our main problem. In this case a single sphere in a 3D periodic computational domain is considered and at very low Reynolds number (Re ≪ 1) simulations are performed for varying diameter (D), and hence varying porosity (e) of the system. We have evaluated the dimensionless drag force, F acting on a single sphere at fully-developed condition, and compared it with semi-analytical results.

\[
F = \frac{F_{f-o}}{3\pi \mu u_s D}
\]

where \(F_{f-o}\) is the total force on the solid body (this includes the force due to viscous shear and pressure force), \(u_s\) represents the superficial velocity and D represents the diameter of the sphere. The superficial velocity \(u_s\) has been calculated by volumetric averaging (including computational cells inside the solid) of the velocity field. The domain is described by a uniform computational grid of 32×32×32 cells with \(\Delta x = 1 \cdot 10^{-3} m\). For all the cases the fluid properties are considered as: \(\rho = 1 \frac{kg}{m^3}, \mu = 0.05 \frac{Pa}{s}\). For each case, a small but finite value of \(\beta\) is assigned such that the flow field remains in the Stokes flow regime (for all the

Table 3. Comparison of the Present Computation of Fluctuation Lift Coefficient \((C_L)\), RMS-Averaged Lift Coefficient \((C_{L, rms})\) and Non-Dimensional Vortex Sheding Frequency \((St)\) with Other Numerical and Experimental Results for Flow Past a Cylinder

<table>
<thead>
<tr>
<th>Re</th>
<th>References</th>
<th>(C_L)</th>
<th>(C_{L, rms})</th>
<th>(St)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100</td>
<td>Present</td>
<td>±0.342</td>
<td>0.238</td>
<td>0.171</td>
</tr>
<tr>
<td></td>
<td>Uhlmann(56)</td>
<td>±0.339</td>
<td>–</td>
<td>0.169</td>
</tr>
<tr>
<td></td>
<td>Tseng and Ferziger(51)</td>
<td>±0.339</td>
<td>0.29</td>
<td>0.164</td>
</tr>
<tr>
<td></td>
<td>Liu et al.(68)</td>
<td>±0.339</td>
<td>–</td>
<td>0.164</td>
</tr>
<tr>
<td>200</td>
<td>Present</td>
<td>±0.683</td>
<td>0.0714</td>
<td>0.198</td>
</tr>
<tr>
<td></td>
<td>Liu et al.(68)</td>
<td>±0.69</td>
<td>–</td>
<td>0.192</td>
</tr>
</tbody>
</table>
The non-dimensional drag forces as a function of the porosity \( \varepsilon \) is found to be about 2.6\% for the most dilute system. Like the other present cases \( Re \approx 0.3 \). In Table 7 the calculated non-dimensional drag forces as function of the porosity \( \varepsilon \) are compared with semi-analytical results.\(^7\) It is found that the maximum deviation is about 2.6\% for the most dilute system. Like the other test cases \( F_{f_{-s}} \) is calculated by the surface integral, consisting of forces due to viscous shear \( F_{\mu} \) and pressure variation \( F_{P} \). Since here we are solving for \( P \) (periodic pressure) instead of \( p \) (actual pressure), the numerically calculated pressure force \( F_{P} \) by the code only contains the force exerted due to the local pressure variation. In the actual system due to the global pressure gradient it also experiences a buoyancy-type force \( F_{B} \) and from the force balance we can write

\[
\vec{\beta} = \frac{F_{f_{-s}}}{V} = \frac{1}{V} \left( F_{\mu} + F_{P} + F_{B} \right) = \frac{1}{V} \left( F_{\mu} + F_{P} + \vec{\beta} V_{S} \right)
\]

\[
= \frac{1}{V} (F_{\mu} + F_{P}) + (1 - \varepsilon) \vec{\beta}
\]

(25)

where \( V \) is total volume of the system and \( V_{S} \) is the volume of the solid particle. Hence, the porosity is \( \varepsilon = 1 - \frac{V_{S}}{V} \). From Eq. 25, we obtain

\[
\vec{\beta} = \frac{F_{\mu} + F_{P}}{\varepsilon V}
\]

(26)

Equations 26 or 27 can be used to calculate \( F_{f_{-s}} \). In all the cases, the left hand side of Eq. 26 (input) and the calculated right hand side of Eq. 26 deviate less than 0.5\%, which shows that a very good overall force or momentum balance is achieved from the simulation method. In Table 7, the grid convergence studies for most dilute system has been shown, which indicates that the current grid resolution is sufficient.

At finite \( Re \), simulation has been performed for \( \varepsilon = 0.592 \) and \( \varepsilon = 0.799 \) in a domain of \( 64 \times 64 \times 64 \) cells. Flow is considered only along the [100] direction of the periodic box. In Figure 9, the computed drag force has been compared with LBM based numerical study by Hill et al.\(^2\) and IBM based numerical result by Tenneti et al.\(^3\) Again a good agreement has been observed.

The spatial (grid) convergence of non-dimensional drag has been examined for \( \varepsilon = 0.8869 \) at \( Re \approx 35 \). The simulation has been performed for total five different grids with \( D/\Delta x = 7.2 \), 15, 30, 60, and 192 by applying equal pressure gradient. The calculated drag force using \( D/\Delta x = 192 \) is considered as resolution free result to compute the errors for other grids. In Figure 10 the \% error has been plotted with dimensionless grid resolution \( D/\Delta x \) and a second order convergence (global accuracy) is clearly observed.

**Foam geometry and simulation set-up**

After developing an accurate numerical tool the next task is to represent the foam unit-cell geometry in the computational domain. As mentioned earlier in this current work a tetrakaidecahedron (Kelvin’s unit cell) is considered as idealized representative unit-cell of foam in a periodic domain capturing the relevant characteristics of the flow without losing any generalities. A unit tetrakaidecahedron shape is shown in Figure 11 which has a total of 6 square and 8 hexagonal holes, its sides being formed by intersecting spherocylinders. Commercially metallic foams are described by two quantities: the porosity and the pore density (typically expressed by pores-per-inch: PPI), which is basically a function of the ligament diameter \( d_{l} \) and the length of the unit periodic cell \( L_{p} \) shown in Figure 11. \( L_{e} \) represents the length of each ligament, which is equal to \( L_{p}/2\sqrt{2} \) and equal for all the sides. For all the simulation cases the value of \( L_{p} \) is fixed and a set of twelve different foams of different porosity \( \varepsilon = 0.638–0.962 \) are obtained by changing the \( d_{l}/L_{p} \) ratio. For flow through porous media there exists a large variety of definitions for the length scale. In this work we define the relevant length scale/hydraulic diameter based on an equivalent spherical diameter as

\[
d_{eq} = \frac{6V_{S}}{A_{S}}
\]

(28)

where \( V_{S} \) and \( A_{S} \) represent the solid volume and solid surface area in a single periodic domain, both are calculated numerically from volume and surface integrals. The different geometrical parameters are tabulated in Table 8. The Flow field is generated by applying a constant body force or pressure gradient \( \vec{\beta} \) across the periodic faces and the main drawback of the current approach is that the value of \( Re \) is unknown prior to the simulation. To set \( Re \) beforehand, we have performed a few extra sets of simulations on a coarse grid to obtain an approximate Ergun type correlation relating the pressure drop with \( Re \). With the help of this approximate correlation we
could estimate proper values of $\tilde{\beta}$ for the different \( Re \) numbers of interest.

To analyze the resulting drag force we use the viscosity to non-dimensionalize the drag or pressure gradient

\[ f = \frac{\Delta p}{\rho u_s^2} \]  

(29)

where \( u_s \) is the superficial or empty volume velocity, that is also used to define Reynolds number

\[ Re = \frac{\rho u_s d_{eqv}}{\mu} \]  

(30)

It is very common to write the pressure loss in a porous medium as the sum of two terms: viscous drag due to the presence of the solid wall, proportional to fluid velocity and viscosity, and form drag, proportional to the density and velocity squared. For engineering purposes, the most well-known formula for predicting the pressure drop in a randomly packed bed of particles is the Ergun correlation. In this analysis we

**Table 4. The Variation of Non-Dimensional Length of Recirculation Zone \( (L_w/D) \) with \( Re \) for Flow Past a Cylinder**

<table>
<thead>
<tr>
<th>( Re )</th>
<th>( L_w/D ) (Present)</th>
<th>Takami et al.(^{63}) (Numerical)</th>
<th>Tseng and Ferziger(^{51}) (Numerical)</th>
<th>Ye et al.(^{59}) (Numerical)</th>
<th>Silva et al.(^{64}) (Numerical)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>0.88</td>
<td>0.932</td>
<td>–</td>
<td>0.92</td>
<td>1.05</td>
</tr>
<tr>
<td>40</td>
<td>2.20</td>
<td>2.325</td>
<td>2.21</td>
<td>2.27</td>
<td>2.54</td>
</tr>
<tr>
<td>50</td>
<td>2.91</td>
<td>3.011</td>
<td>–</td>
<td>–</td>
<td>2.78</td>
</tr>
</tbody>
</table>

**Table 5. Simulation Settings for Flow Past a Sphere**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Delta x=\Delta y=\Delta z )</td>
<td>( G10: 2 \cdot 10^{-2}, G20: 1 \cdot 10^{-2}, G30: 6.67 \cdot 10^{-3} )</td>
</tr>
<tr>
<td>Diameter (( D ))</td>
<td>0.2</td>
</tr>
<tr>
<td>Viscosity</td>
<td>0.1</td>
</tr>
<tr>
<td>Density</td>
<td>5, 10, 15, 20, 25, 50, 75, 150, 175, 250</td>
</tr>
<tr>
<td>( U_{in} )</td>
<td>1</td>
</tr>
<tr>
<td>Time step</td>
<td>1 \cdot 10^{-3}</td>
</tr>
</tbody>
</table>
start our analysis from the ansatz that the pressure drop can also be written in this form

$$\frac{\Delta p}{L} = a \frac{\mu}{d_{eq}} u_s + b \frac{p}{d_{eq}} u_s^2$$  \hspace{1cm} (31)$$

which, using Eq. 29, can also be written as

$$f = a + b Re$$  \hspace{1cm} (32)$$

One of the main advantages of the current non-dimensional form is that the viscous and inertial contribution can be easily considered separately. At the same time there is no need to numerically calculate the drag force (using a surface integral). Instead, only the superficial fluid velocity along with the input \( \nabla p \) (= \( \vec{f} \)) are required.

Starting from creeping flow, the \( Re \) is varied up to \( \sim 500 \) and in this range in total ten values of \( Re \) are considered for foams of 12 different porosities. We have considered four extreme situations, lowest and highest \( Re \) case for most dilute and most packed system to check the grid dependency (see Appendix). We found that a grid resolution of 200 \( \times 200 \times 200 \) provides grid independent results for these extreme cases and hence we used these settings for the entire range of \( \varepsilon \) and \( Re \). The time-step is set on basis of the Courant number, allowing a maximum value of \( CFL \approx 0.4 \). Starting from an initial zero velocity field, the physical time to reach steady state is inversely-proportional to the kinematic viscosity (\( \nu = \mu/\rho \)) for a fixed solid structure.\(^{20}\) To decrease the total physical time to reach a steady state (or quasi-steady state for high \( Re \) cases) the value of \( \nu \) should be low and at the same time if we use very low values of \( \nu \), to maintain a specific \( Re \) the velocity should be high, which demands smaller time-steps for proper temporal convergence. For this large parameter space it is very important to select the proper fluid properties to optimize the total computational time. We have used \( \mu = 0.003 \) Pa s for all the cases, and \( \rho = 20 \text{ kg/m}^3 \) for \( Re \approx 1, 2, \) and 5, \( \rho = 150 \text{ kg/m}^3 \) for \( Re \approx 10, 20, \) and 50 and \( \rho = 1000 \text{ kg/m}^3 \) for \( Re \approx 100, 200, \) and 500; and as a result for all the cases the time-step required to satisfy the CFL criteria remains \( \Delta t \sim O \left( 10^{-5} \right) \) and the total time to reach a (quasi) steady state is found \( t_{\text{total}} \sim O(1) \). For the cases with finite \( Re \), besides \( \varepsilon \) and \( Re \) the total drag force also depends on the flow direction, due to the dominance of the inertial contribution. For varying flow direction the fluid experiences different flow openings, which causes variation in pressure drop. To change the flow direction, the direction of the applied pressure drop or body force (\( \vec{f} \)) has been changed and in total three cases with lattice-direction (\( \Theta \)) equal to \{100\}, \{110\}, and \{111\} have been considered. Figure 12 illustrates these three cases in a periodic domain and the final drag correlation is proposed by taking the linear average of these three cases.

### Results and Discussions

#### Stokes flows

At very low Reynolds number, i.e., for Stokes flow through a porous matrix, the flow is governed by Darcy’s law that links the 1-D pressure drop (\( \Delta p \)) with the superficial velocity (\( u_s \)) as

$$\frac{\Delta p}{L} = \frac{\mu}{\kappa} u_s$$  \hspace{1cm} (33)$$

Equation 33 is a very general relation for pressure drop over a porous medium where \( \kappa \) represents the permeability, which is a macroscopic parameter that depends on the geometrical details of the porous media. One of the most widely accepted

### Table 6. Comparison of the Present Computation of Drag Coefficient (\( C_d \)) with Other Numerical and Experimental Results for Flow Past a Sphere

<table>
<thead>
<tr>
<th>( Re )</th>
<th>Domain size</th>
<th>( C_d )</th>
<th>Present ( (\text{Experimental: curve fit}) )</th>
<th>Clift et al.(^{71})</th>
<th>Mittal et al.(^{53})</th>
<th>Johnson and Patel(^{69})</th>
<th>Marella et al.(^{70})</th>
</tr>
</thead>
<tbody>
<tr>
<td>10</td>
<td>20D ( \times 20D \times 20D )</td>
<td>4.195</td>
<td>4.383</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>20</td>
<td>20D ( \times 20D \times 20D )</td>
<td>2.642</td>
<td>2.735</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>30</td>
<td>20D ( \times 20D \times 20D )</td>
<td>2.06</td>
<td>2.122</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>40</td>
<td>20D ( \times 20D \times 20D )</td>
<td>1.739</td>
<td>1.788</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>50</td>
<td>20D ( \times 20D \times 20D )</td>
<td>1.527</td>
<td>1.574</td>
<td>–</td>
<td>1.57</td>
<td>1.56</td>
<td>–</td>
</tr>
<tr>
<td>100</td>
<td>20D ( \times 20D \times 20D )</td>
<td>G20 : 1.065</td>
<td>1.087</td>
<td>1.08</td>
<td>1.08</td>
<td>1.06</td>
<td>–</td>
</tr>
<tr>
<td>100</td>
<td>1D ( \times 8D \times 8D )</td>
<td>G10 : 1.095</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G20 : 1.0861</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G40 : 1.0594</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G60 : 1.0592</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>150</td>
<td>1D ( \times 8D \times 8D )</td>
<td>0.880</td>
<td>0.889</td>
<td>0.88</td>
<td>0.90</td>
<td>0.85</td>
<td>–</td>
</tr>
<tr>
<td>300</td>
<td>1D ( \times 8D \times 8D )</td>
<td>0.648</td>
<td>0.653</td>
<td>0.68</td>
<td>0.629</td>
<td>0.621</td>
<td>–</td>
</tr>
<tr>
<td>350</td>
<td>1D ( \times 8D \times 8D )</td>
<td>0.620</td>
<td>0.619</td>
<td>0.63</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td>500</td>
<td>1D ( \times 8D \times 8D )</td>
<td>G10 : 0.6060</td>
<td>0.555</td>
<td>–</td>
<td>–</td>
<td>–</td>
<td>–</td>
</tr>
<tr>
<td></td>
<td>G20 : 0.5658</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>G30 : 0.5657</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Unless specified otherwise all reported data are obtained with the G20 grid.
The generalized form of permeability is the Carman-Kozeny (CK) approximation
\[ \kappa = \frac{r_h^2}{S T^2} \] (34)
where \( \varepsilon \) is the porosity, \( S \) is a shape factor that depends on the geometrical shape of the porous medium, \( T \) is the hydraulic tortuosity, which represents the effective elongation of the fluid path in the porous medium, and \( r_h \) is the hydraulic radius, defined as the ratio of the free volume to the wetted surface area. Sometimes, \( S T^2 \) is called the Kozeny constant \( (k) \), and taken equal to 5, independent of the geometry. Later it was found that the value of \( k \) can be as high as 50 depending on the shape of the porous medium. For a porous medium defined by a random configuration of mono-disperse spheres
\[ r_h = \frac{\varepsilon}{6(1-\varepsilon)} D \] (35)
where \( D \) is the diameter of the spheres. Hence, in the current non-dimensional form the Carman-Kozeny (CK) correlation reads
\[ a = \frac{f_{\text{Stokes,CK}}}{180} \left(1-\frac{\varepsilon}{\varepsilon^3}\right)^2 \] (36)
Perhaps, the most well-known formula to predict the pressure drop in randomly packed beds is the Ergun correlation which for \( Re \to 0 \) corresponds to
\[ a = \frac{f_{\text{Stokes,Ergun}}}{150} \left(1-\frac{\varepsilon}{\varepsilon^3}\right)^2 \] (37)
Universal values and/or form of the CK and Ergun constants have been a subject of considerable debate. Several studies on packed beds filled with particles with different shape and size suggested that actually an Ergun-type of pressure drop correlation ought to be determined empirically for each configuration. For the present porous media made of open-cell solid foams, to calculate \( f_{\text{Stokes}} \) we have applied a pressure drop across the periodic faces such that for all the cases \( Re < 0.01 \). The non-dimensional drag has been plotted vs. solids fraction \( (1-\varepsilon) \) in Figure 13 along with the CK and Ergun correlations for the entire range of the porosity \( (\varepsilon) \). With increase in the total solids volume fraction \( (1-\varepsilon) \), the total drag force increases exponentially due to the rapid increase in solid surface area per unit volume of the system. A few additional simulations have been carried out to check the effect of the flow direction on the friction factor in this low \( Re \) regime. As expected, we found almost no deviation \((\sim 0.02\%)\) with \( \Theta \) as in the Stokes flow the inertial terms in the N-S are negligible and at steady state the applied pressure drop is only balanced by the wall friction, that is proportional to the available surface area and invariant with respect to \( \Theta \). The slight deviation is due to numerical effects, for a same setup with change in flow direction the effective grid resolution also slightly changes. A few more simulations have been performed for a foam of the same

<table>
<thead>
<tr>
<th>Porosity (( \varepsilon ))</th>
<th>Grid Resolution ( (D/\Delta x) )</th>
<th>( F_{\text{Analytical}} )</th>
<th>( F_{\text{IBM}} )</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.4764</td>
<td>32.0</td>
<td>42.14</td>
<td>42.01</td>
<td>0.31</td>
</tr>
<tr>
<td>0.5511</td>
<td>30.4</td>
<td>28.10</td>
<td>27.82</td>
<td>0.99</td>
</tr>
<tr>
<td>0.6567</td>
<td>27.8</td>
<td>15.40</td>
<td>15.48</td>
<td>-0.52</td>
</tr>
<tr>
<td>0.7845</td>
<td>23.8</td>
<td>7.44</td>
<td>7.46</td>
<td>-0.27</td>
</tr>
<tr>
<td>0.8774</td>
<td>19.9</td>
<td>4.29</td>
<td>4.33</td>
<td>-0.93</td>
</tr>
<tr>
<td>0.9358</td>
<td>15.9</td>
<td>2.81</td>
<td>2.82</td>
<td>-0.36</td>
</tr>
<tr>
<td>0.9731</td>
<td>11.9</td>
<td>2.008</td>
<td>1.965</td>
<td>2.40</td>
</tr>
<tr>
<td></td>
<td>23.8</td>
<td>1.971</td>
<td>1.84</td>
<td></td>
</tr>
<tr>
<td></td>
<td>47.6</td>
<td>1.975</td>
<td>1.64</td>
<td></td>
</tr>
<tr>
<td></td>
<td>95.2</td>
<td>1.973</td>
<td>1.74</td>
<td></td>
</tr>
</tbody>
</table>

Table 7. Comparison of Non-Dimensional Drag Force \( (F) \) for Simple Cubic Array of Sphere at Stokes Flow Regime with Varying Porosity \( (\varepsilon) \).
porosity, but with \( Re \sim 0.05, Re \sim 0.1, \) and \( Re \sim 0.2. \) The extrapolation of the drag value to \( Re = 0 \) is invariant with the present results for \( Re \sim 0.01. \) This suggests that the simulations are performed at a sufficiently small Reynolds number. To fit the simulation data we use a correlation with a similar form as the CK and Ergun correlations, using however different constants. The fit obtained from nonlinear regression is given by

\[
f_{\text{Stokes}} = 70.54 \left(1 - \varepsilon\right)^{1.38} \varepsilon^{1.5}
\]

Figure 13 shows that for creeping flows, the correlation of the same form as CK or Ergun \( f = p(1 - \varepsilon)^{1/2}/\varepsilon \) with different constants describes the simulation data very accurately. The maximum deviation of this fit is less than 0.7% where the average deviation is only 0.35%. In all the cases, both the Ergun and CK correlations, under-predict the pressure drop for solid foams. Similar behavior has been observed by Nemec et al.\(^74\) and Dorai et al.\(^75\) for a porous medium made of randomly packed cylinders. This is not very surprising, since the Ergun correlation is derived from experimental studies of randomly packed spheres in the closely packed limit (porosity in the range of 0.4–0.7), whereas in the present case both the porosity limit and the internal structures are different.

**Finite Reynolds-number flows**

When the Reynolds number is gradually increased a nonlinear term appears in the expression of the pressure gradient (Eq. 32); this is known as the so called transition from the Darcy flow regime to the Forchheimer flow regime. In the Darcy flow regime the fluid inertia is neglected, whereas as in the Forchheimer flow regime it is pronounced. The exact limiting value of this transition depends on the structure of the porous medium and definition of the \( Re \). With increasing \( Re \), the microscopic inertial forces related to the local acceleration of the fluid particles increase. This inertial contribution in the overall pressure drop in porous media is expressed by a \( u_i^2 \) term by almost all authors, and can be justified by the time-integration of the average 1D macroscopic advection term \( \left(u_i, \nabla u_i\right) \). In this transition regime the detailed visualization of the flow not only helps to interpret the numerical results (the dependency of the drag force on \( Re, \varepsilon, \) and \( \Theta \)), but also helps to explain how the flow field is modified by the fluid inertia and how it changes the flow features during the transition from the linear to the nonlinear regime.

Table 8. Geometrical Parameters of Foam Unit-Cell Used in this Study

<table>
<thead>
<tr>
<th>Porosity (( \varepsilon ))</th>
<th>( L_p ) (mm)</th>
<th>( L_i/d_s )</th>
<th>( d_s/\Delta x )</th>
<th>( d_{app} ) (mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.962</td>
<td>10</td>
<td>4.50</td>
<td>15.7</td>
<td>1.257</td>
</tr>
<tr>
<td>0.953</td>
<td>10</td>
<td>4.00</td>
<td>17.7</td>
<td>1.420</td>
</tr>
<tr>
<td>0.940</td>
<td>10</td>
<td>3.50</td>
<td>20.2</td>
<td>1.637</td>
</tr>
<tr>
<td>0.931</td>
<td>10</td>
<td>3.25</td>
<td>21.8</td>
<td>1.771</td>
</tr>
<tr>
<td>0.920</td>
<td>10</td>
<td>3.00</td>
<td>23.6</td>
<td>1.929</td>
</tr>
<tr>
<td>0.906</td>
<td>10</td>
<td>2.75</td>
<td>25.7</td>
<td>2.123</td>
</tr>
<tr>
<td>0.864</td>
<td>10</td>
<td>2.25</td>
<td>31.4</td>
<td>2.668</td>
</tr>
<tr>
<td>0.813</td>
<td>10</td>
<td>1.875</td>
<td>37.7</td>
<td>3.291</td>
</tr>
<tr>
<td>0.762</td>
<td>10</td>
<td>1.625</td>
<td>43.5</td>
<td>3.918</td>
</tr>
<tr>
<td>0.728</td>
<td>10</td>
<td>1.50</td>
<td>47.1</td>
<td>4.344</td>
</tr>
<tr>
<td>0.687</td>
<td>10</td>
<td>1.375</td>
<td>51.4</td>
<td>4.882</td>
</tr>
<tr>
<td>0.638</td>
<td>10</td>
<td>1.25</td>
<td>56.6</td>
<td>5.563</td>
</tr>
</tbody>
</table>
away from the solid walls. Figure 14a reveals a similar behaviour. With increasing Re, the areas with the maximum velocity contours are located further downstream and have a lower strength (Figure 14b, c)—this indicates the pronounced effect of boundary layers being formed at the solid walls and the onset of the nonlinear inertial contribution in the macroscopic flow features. With further increase in Re, a jet like feature appears in the parts with the smallest open area available for flow (Figure 14d, e). This is very similar to the inviscid core formed in channel flows. Dybbs and Edwards experimentally reported a similar flow feature in porous media made of spherical and cylindrical packings. They termed this jet-like feature the “inertial core” and identified this as the main source for the nonlinear relationship between pressure drop and flow rate. The present numerical results suggest that this is indeed the case. The length of this inertial core further increases with increasing Re (Figure 14 f, g) and starts to touch the core of the next periodic cell, creating a seamless jet of fluid in the main flow direction. For this particular porosity, the streamlines for Re ~ 20 or higher shows the presence of a recirculation zone at the solid surface. With increasing Re, the volume of the recirculation zone increases and the center of recirculation zone shifts further downstream, which is very similar to flow past a body in an unconfined flow domain. However, here due to the presence of another solid object downstream, the strength of the recirculation is less compared to the case where a solid body is placed in an unconfined domain. For this particular case, unsteady behavior is first observed at Re ~ 100 in the form of a small oscillation of the inertial core. At Re ~ 200 the flow becomes highly unsteady, the stable inertial core breaks down and the vigorous fluid motion leads to good mixing throughout the domain. Similar behavior is observed for all other cases (different ε and Θ), although the onset of the first unsteady flow features is found at comparatively higher Re for higher solids volume fraction (1−ε). The associated increase of the specific surface area in the fixed system restricts the micro-vortices to form and to be carried downstream. As a result, the macroscopic strength of the unsteady behavior also decreases with increase in solids volume fraction.

For the current definition of Re, for a fixed Θ, it is interesting to observe that for larger solids volume fractions the Darcy regime spans a comparatively large range of Re. For instance, consider two extreme cases: ε=0.962 and ε=0.638. If we compute the ratio f/Stokes when the flow is directed along Θ={1 0 0}, we found that for ε=0.962 at Re ~ 2.4, f/Stokes = 1.21 and for ε=0.638 at Re ~ 10, f/Stokes = 1.17. The deviations of f/Stokes from unity indicates the contribution of the fluid inertia and we find a similar magnitude of f/Stokes at higher Re for closely packed systems. Figure 15 shows the velocity contours and stream lines for ε=0.728, Θ={1 1 0} with varying Re. Similar to Figure 14, the streamlines are 2D. A more detailed comparison of Figure 15 with Figure 14 leads to two main conclusions. First of all, when the flow is directed along Θ={1 1 0} the internal complex geometry allows to form two inertial cores, however these inviscid cores are able to extend further for higher Re due to the presence of a solid ligament (perpendicular to flow direction) in the downstream direction. As a result, the flow experiences a higher pressure.
drop compared to $\Theta = \{100\}$. The second conclusion is that the inertial core forms at higher $Re$ for more dense systems. It indicates that at high solids volume fractions the transition from the Darcy to the Forchheimer flow regime occurs at higher $Re$, which is, presumably, due to the higher effective characteristic length scale (Eq. 28) at higher solids volume fraction.

Figure 16 illustrates the simulation results for $\varepsilon=0.962$ and $H=100$ for different flow angles ($\Theta$). In sub-figures (a), (c), and (e) the indicated planes are parallel to the main flow direction. In each of these planes the streamlines and resultant velocity contours are shown. Due to the symmetric nature of the flow in these mid-planes the velocity component perpendicular to the plane is zero. Conversely, sub-figures (b), (d), and (f) show the velocity contours for the mid-plane perpendicular to the flow direction. In sub-figures (a), (c), and (e) the different size and shape of the inertial core at the same $Re$ are clearly visible. It also shows the different flow openings depending on the flow direction. At high $Re$ this is the main cause for attaining a different pressure drop at the same $Re$. This will be discussed quantitatively in the next paragraphs.

For the unsteady flows, the drag force fluctuates over time. Here, we report the time average values obtained after the flow reaches a quasi-steady state. The magnitude of the fluctuations in the non-dimensional drag force calculated based on the surface integral of the viscous and pressure forces, is higher compared to that based on the superficial velocity and applied body force. The second method inherently uses
volumetric averaging which suppresses the additional fluctuation. Although both calculations, averaged over a large span of time, provide the same non-dimensional drag force within numerical accuracy, here we use the second method. As a result a comparatively smaller time-history can be used in the time averaging. With $Re$, the magnitude of the fluctuations increases, reaching a maximum after which it decreases again. When the flow is directed along $H$, it becomes unsteady at relatively lower $Re$ where the magnitude of fluctuation is found to be relatively high as compared to the case with $\Theta = \{1 0 0 \}$. Most probably this is due to the fact that in the latter case the fluid finds a relatively straight trajectory compared to other cases.

The creeping flow results, presented in the previous section clearly point out that, in Eq. 31, $a$ is only a function of porosity. It is a very common observation for all porous media and well documented in the literature. At the same time, the Ergun correlation, developed based on experiments for random packings of spheres at moderate and high $Re$, suggests that $b=F(\varepsilon)$ where a constant value of $b$ is valid for the entire range of $Re$ for a particular packing. However, the numerical results by Hill et al.\textsuperscript{a} for ordered and random arrays of spheres shows that the linear dependency of $f$ on $Re$ occurs only at large values of $Re$, although the slope of the $f$ vs. $Re$ curve changes with flow angle ($\Theta$). By performing experiments on a typical metal foam Dukhan et al.\textsuperscript{18} proposed a fit for $f$ with $Re$ by a piecewise linear function, with changing gradient. In this study the dependency of $Re$ on $f$ is obtained numerically with special attention at the low $Re$ regime. Figures 17 and 18 show the variation of the non-dimensional friction factor ($f$) with varying $Re$ for different porosities ($\varepsilon$) and different flow angles ($\Theta$). Sub-figure (a), (c), and (e) are for low to moderate $Re$ flows and sub-figure (b), (d), and (f) for the high $Re$ cases. In the sub-figure (a), (c), and (e), the initial parts of the $f$ vs. $Re$ curves are flat, which reflects the presence of the Darcy regime. However, the slope of the curve in the post-Darcy regime changes. In most reported studies, it was tried to link the full $f$ vs $Re$ curve by a single curve with a single slope. However, this procedure gives difficulties and large deviations in the low to moderate $Re$ region.\textsuperscript{18} When approximately $Re > 20$, sub-figure (b), (d), and (f) of Figures 17 and 18 show almost linear variation of $f$ with $Re$, at least for the range of $Re$ studied in this work, and the slopes of the $f$ vs. $Re$ curves vary with $\varepsilon$ and $\Theta$, i.e., $b=F(\varepsilon, \Theta)$. Table 9 contains the values of $b$ for different porosities and flow directions obtained by curve fitting at $20 < Re < \sim 500$. It shows that for the entire range of porosity, $b$, and hence $f$, increase rapidly with the solids volume fraction $(1-\varepsilon)$. It is very interesting to observe that for all the three flow directions, similar to the Ergun correlation, $b$ can be fit in the form of $p(1-\varepsilon)^q/\varepsilon^r$, with different constants $(p, q, \text{and} r)$. For $\Theta = \{1 0 0 \}$, $\Theta = \{1 1 0 \}$ and $\Theta = \{1 1 1 \}$ we found $[p, q, r]$ as [3.102, 1.362, 3.279], [2.091, 1.195, 5.056], and [0.637, 0.7824, 5.781], respectively. The maximum error between the fit and discrete data is less than $\sim 6\%$, while the average error is only $\sim 2\%$.

For the entire range of porosities, the slope of $f$ vs. $Re$ curve (i.e., $b$), qualitatively indicates the relative contribution of the inertia in the total drag force, is lower for $\Theta = \{1 0 0 \}$ than for
Similar behavior has been reported by Hill et al.\textsuperscript{20} for flow past ordered arrays of spheres. The total inertial contribution to drag mainly depends on the effective flow opening in the direction of the flow. Figure 16b, d, and f show the plane at the halfway of the unit cell perpendicular to the flow and by closely looking at these images we can conclude that the effective flow area is less for $\Theta = \{1\ 1\ 0\}$ compared to $\Theta = \{1\ 0\ 0\}$. Also for $\Theta = \{1\ 1\ 0\}$ the inertial core gets restricted by another solid ligament in the direction of the flow (Figure 16c). As a result, for $\Theta = \{1\ 1\ 0\}$, presumably, as this effective flow area is less, the fluid flow velocity increases and the flow experiences a higher drag force. However, until $\varepsilon = 0.920$, the values of $b$ for $\Theta = \{1\ 1\ 1\}$ are greater than for both $\Theta = \{1\ 0\ 0\}$ and $\Theta = \{1\ 1\ 0\}$, whereas for higher solids fraction it is less. Again this is due to the effect of available relative flow opening and restriction in flow path. For different porosities, for this complex system it changes in a non-intuitive manner.

**Drag correlation**

In actual flow situations, the orientation of the foam unit cell may not necessarily be aligned along its axes. For engineering application purposes, an average pressure drop or drag correlation is derived from the simulation results as function of $\varepsilon$ and $Re$, where we have averaged the results over for all the three different $\Theta$. For a particular $\varepsilon$, for a specified body force ($\beta$), the generated superficial velocities ($u_s$) are averaged.
for all three Θ directions and the non-dimensional drag is calculated based on the average \( \mu_c \). From the previous discussion it is clear that, for different ranges of \( Re \), the normalized pressure drop, \( f \) has a different dependency on \( Re \). Trying to capture these dependencies in a single representation would lead to large errors. Hence, we have divided the current \( Re \) range in two parts.

For \( Re < 20 \), \( b \) depends on \( Re \) and we propose the non-dimensional drag correlation as

\[
f_1 = 70.54 \left( \frac{1-\varepsilon}{\varepsilon} \right)^{1.38} + 0.84 \left( \frac{1-\varepsilon}{\varepsilon} \right)^{0.59} Re^{0.39+1.46(1-\varepsilon)^{0.51}}
\]  

(39)

This equation fits all the simulation results with a maximum error of 3% and an average error of 1.5%.

For \( Re > 20 \), \( b \) is independent of \( Re \) and we can use a form similar to the Ergun equation

\[
f_2 = 70.54 \left( \frac{1-\varepsilon}{\varepsilon} \right)^{1.38} + 1.95 \left( \frac{1-\varepsilon}{\varepsilon^4.39} \right) Re
\]  

(40)

For this equation the average relative deviation is 2.5% with a maximum deviation of 5%. The inertial contribution on the average non-dimensional drag force, i.e., \( f - f_{Stokes} \), as a function of \( Re \) and \( \varepsilon \) is shown in Figure 19. For proper visualization, the low and moderate \( Re \) cases are shown in sub-figures (a–c), whereas sub-figures (d–f) show the cases with high \( Re \). Eqs. 39 and 40 do not cross at \( Re = 20 \). Therefore, finally we propose a practical continuous correlation based on Eqs. 39 and 40, which is valid for the entire range of \( Re \).

Figure 17. Normalized drag force as a function of \( Re \) for low solids volume fractions (\( \varepsilon = 0.906 - 0.962 \)) for different mean flow directions: (a, b) \( \Theta = \{100\} \), (c, d) \( \Theta = \{110\} \), and (e, f) \( \Theta = \{111\} \).

The sub-plots (a, c, e) are for finite but low \( Re \) flows, where sub-plots (b, d, f) are for the upper range of \( Re \). At moderate and high \( Re \), the slope of \( f \) with \( Re \) is almost constant for a particular porosity. The symbols represent the simulation results, and the continuous blue lines in sub-plots (b, d, f) indicate the lines of constant slope.
for $\varepsilon > 0.8$. $f = \max(f_1, f_2)$ else $f = \min(f_1, f_2)$  

(41)

In Figure 19, the continuous smooth lines are representing the final curve-fit (Eq. 41) where at low to moderate $Re$ the change in slope can been observed. For the entire range of porosities, Eq. 41 uses 40 for large $Re$ cases. Figure 19a–c show the deviation from 39 to the final fit for different values $\varepsilon$. For a few points the maximum deviation between the final fit and simulation results are as high as 12%. However, the average deviation is 5.6%.

Most of the experimental studies available in the literature have concentrated on moderate and high $Re$. Presumably to achieve a better fit at higher $Re$, the accuracy is sacrificed in the lower $Re$ range. Commercially available porous foam geometries are very random, and for highly porous foams the ligament cross-sections are concave triangles. Still it is interesting to compare the present simulation results with existing experimental results for real foam samples. Dietrich et al.3

Table 9. Slope of the $f$ vs. $Re$ Curve, i.e., $b$ for Different Flow Direction ($\Theta$) and Porosity ($\varepsilon$)

<table>
<thead>
<tr>
<th>$\Theta$</th>
<th>$\varepsilon$</th>
<th>0.962</th>
<th>0.953</th>
<th>0.940</th>
<th>0.931</th>
<th>0.920</th>
<th>0.906</th>
<th>0.864</th>
<th>0.813</th>
<th>0.762</th>
<th>0.728</th>
<th>0.687</th>
<th>0.638</th>
</tr>
</thead>
<tbody>
<tr>
<td>{100}</td>
<td>0.0425</td>
<td>0.0578</td>
<td>0.0841</td>
<td>0.1041</td>
<td>0.1319</td>
<td>0.1732</td>
<td>0.2301</td>
<td>0.3301</td>
<td>0.6322</td>
<td>1.1001</td>
<td>1.5155</td>
<td>2.2139</td>
<td>3.5147</td>
</tr>
<tr>
<td>{110}</td>
<td>0.0524</td>
<td>0.0703</td>
<td>0.0998</td>
<td>0.1235</td>
<td>0.1463</td>
<td>0.1940</td>
<td>0.3803</td>
<td>0.7533</td>
<td>1.5082</td>
<td>2.2066</td>
<td>3.4814</td>
<td>6.0398</td>
<td></td>
</tr>
<tr>
<td>{111}</td>
<td>0.0591</td>
<td>0.0781</td>
<td>0.1030</td>
<td>0.1243</td>
<td>0.1490</td>
<td>0.1870</td>
<td>0.3217</td>
<td>0.6010</td>
<td>1.0571</td>
<td>1.4813</td>
<td>2.1930</td>
<td>3.4441</td>
<td></td>
</tr>
</tbody>
</table>
reviewed experimental pressure drop correlation/values from about twenty authors including around one hundred different foam samples (0.777 < \varepsilon < 0.978) and proposed a pressure drop correlation. It consists of more than 2500 experimental values and a large number of data points lie within an error range of \pm 40% of the proposed pressure drop correlation. Hence, this is a good candidate to check our present results. Written in our present non-dimensional form it reads

\[ f_{\text{Dietrich}} = 247.5 \left( \frac{1-\varepsilon}{\varepsilon^3} \right)^{2} + 2.175 \left( \frac{1-\varepsilon}{\varepsilon^3} \right) Re \]  \hspace{1cm} (42)

Figure 19 compares the drag obtained by Eq. 41 to the experimental correlation by Dietrich et al.\textsuperscript{3} for \(Re=5, 10, 20, 50, 100, 200\), and 500 with \(\varepsilon=0.962, 0.94, 0.92, 0.864\), and 0.813 (in the same porosity range) in a log–log plot. It shows that a large number of points lie inside the error range of the experimental correlation.

**Conclusions**

In this work we numerically studied the flow in open-cell foams. The flow is described by a sharp, implicit and second order accurate IBM. The implementation details along with detailed verification studies have been presented. The current computational technique avoids the generation of complex unstructured grids and developing a flow solver for complex domains. Idealized unit-cells of foam in a periodic Cartesian computational domain were considered and the geometry of the unit-cell is approximated based on the structural packing...
of a tetrakaidecahedron (Kelvin’s unit cell) with a cylindrical strut morphology. In total, 336 simulations have been performed for different porosities (∏ ≥ 0.962), Reynolds numbers (0 < Re < 500) and flow directions (Θ). Through this extensive parameter study, we are able to very closely show how the drag force and flow features change depending on the different parameters.

For the first time, the means of an accurate numerical model the transition in the flow features from the Darcy to the inertial core forms, which is the main reason for the nonlinear behavior between velocity and pressure drop. The extension of the Darcy flow region depends on the porosity of the foam and extends to relatively higher Re for denser packed system. For the case of denser packed systems the velocity fluctuations decrease, because of the increased specific area that suppresses the formation of micro vortices.

The drag force on the solid walls was found to change with flow direction, whereas in the absence of fluid inertia (creeping flow) the drag force is independent of the flow direction. For creeping flow the overall pressure drop is balanced by the wall friction, which is a function of available surface area and independent of flow direction. However, when the fluid inertia becomes prominent, the effective cross-sectional area available for fluid flow varies with the flow direction. Consequently, the drag force changes with flow direction. When the effective cross-sectional area decreases, the fluid velocity increases and the flow experiences a large drag force. For the first time, the effect of flow direction has been studied for idealized foam unit-cells.

At creeping flow (Re ≪ 1), the Ergun correlation under predicts the non-dimensional drag force. This is not very surprising, as the Ergun correlation was obtained from experimental data for randomly packed spheres (high solids volume fraction) and probably to match the drag at higher Re, it sacrificed accuracy in the low Re regime. It is interesting to find that for the creeping flow regime a drag correlation with the same form as the Ergun equation fits the numerical results very nicely. When the fluid inertia dominates (for Re > 20), the drag force scales linearly with Re. However, in the intermediate “post-Darcy” range 0 ≤ Re < 20 the drag force has a nonlinear dependency on Re, which agrees with observations made by Hill et al.20 for flow past ordered arrays of spheres.

Finally, we proposed a non-dimensional drag force in terms of ∏ and Re, by averaging the drag forces over all the three studied flow directions. The maximum deviation between the final fit and discrete simulation results is 12% (few cases) and the average deviation is 5.6%. The proposed drag correlation is also compared with an empirical correlation. We found that for moderate and high Re cases the present results are within the accuracy range of the experiments, as most of the experiments concentrated on high Re.

### Literature Cited

APPENDIX: GRID INDEPENDENT STUDY FOR FOAM GEOMETRY

Grid convergence studies have been performed for four extreme cases $\varepsilon=0.638$, $Re=0$, $\varepsilon=0.962$, $Re=0$, $\varepsilon=0.638$, $Re \sim 500$ and $\varepsilon=0.962$, $Re \sim 500$ where flow is directed along $\Theta=[1,0,0]$. Three different grid resolutions, $G_1 (100\times100\times100)$, $G_2 (200\times200\times200)$, and $G_3 (320\times320\times320)$ are chosen for all the cases. The same fluid properties and same $\beta$ have been used for all the grids, however the time-step is chosen based on the CFL criterion. The drag force at infinite grid resolution ($G_{inf}$) is estimated by second order Richardson Extrapolation. Table 10 lists the calculated drag force for all the cases along with the relative error compared to the $G_{inf}$ grid. It shows that $G_2$ provides reasonable grid independent results, which is therefore used for all the other cases.

Manuscript received Nov. 15, 2015, and revision received July 19, 2016.