Direct numerical simulation for flow and heat transfer through random open-cell solid foams

Citation for published version (APA):

Das, S., Deen, N. G., & Kuipers, J. A. M. (2016). Direct numerical simulation for flow and heat transfer through random open-cell solid foams: Development of an IBM based CFD model. *Catalysis Today*, 273, 140-150. <https://doi.org/10.1016/j.cattod.2016.03.048>

Document license:
TAVERNE

DOI:
[10.1016/j.cattod.2016.03.048](https://doi.org/10.1016/j.cattod.2016.03.048)

Document status and date:
Published: 15/09/2016

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.

Direct numerical simulation for flow and heat transfer through random open-cell solid foams: Development of an IBM based CFD model

Saurish Das^a, Niels G. Deen^{a,b,*}, J.A.M. Kuipers^a

^a Multiphase Reactors Group, Department of Chemical Engineering and Chemistry, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

^b Multiphase and Reactive Flows Group, Department of Mechanical Engineering, Eindhoven University of Technology, P.O. Box 513, 5600 MB Eindhoven, The Netherlands

ARTICLE INFO

Article history:

Received 14 December 2015

Received in revised form 18 February 2016

Accepted 21 March 2016

Available online 12 April 2016

Keywords:

Porous medium

Open-cell solid foam

Pore-scale level numerical simulation

Immersed boundary method (IBM)

Conjugate heat transfer

ABSTRACT

In the present contribution a sharp interface Immersed Boundary Method (IBM) for flow and heat transfer has been developed to fully resolve highly complex random solid structure on a non-body fitted Cartesian computational grid. A 3D image data set from a Micro-CT (computed tomography) scan of an actual foam geometry is usually converted into a surface mesh of unstructured triangular elements and the current frame work can embed it as an immersed boundary in the CFD domain. A second-order implicit (semi-implicit) method is used to incorporate fluid-solid coupling for flow (heat transfer) at the non-conforming immersed boundary in a structured grid. Both temperature Dirichlet and Neumann boundary conditions, as well as a conjugate heat transfer condition are incorporated at the fluid–solid interface. The proposed method is validated rigorously with existing literature results. The use of a Cartesian grid for flow makes this method robust, computational friendly, and at the same time it avoids the tedious volumetric mesh generation process for very complex shapes. To demonstrate the capability of the current method, it is applied to study flow and heat transfer of a typical foam sample with different flow properties. A closure for pressure drop and heat transfer coefficient are derived for the typical foam sample.

© 2016 Elsevier B.V. All rights reserved.

1. Introduction

Traditionally packed bed reactors are randomly filled with particles coated with a thin porous layer containing catalyst. The catalyst inside this layer (washcoat) is accessible only via diffusion through the very fine pores and it drives the design towards smaller particles to obtain a larger specific surface area. At the same time the use of very small particles leads to a large pressure drop and one well known solution to overcome this issue, is to apply slurry reactors that allow for very small particles with minimal pressure drop. Another potential option is to use comparatively large pellets consisting of open structures (open-cell solid foams) where washcoat layers are deposited at the outer surface of the pores inside the solid foam structures. To design and optimize such processes at a large industrial scale with feasible computational cost it is necessary to adopt so called multiscale modeling approach and this is

schematically shown in Fig. 1. At the smallest level direct numerical simulation (DNS) are used to fully resolved the pore structures (*micro-scale* or *pore-scale level*) of a small section of solid foam (representative elementary volume, REV), which provides ‘closure free’ detailed flow and heat transfer characteristics. The results from DNS studies can be used to develop momentum and heat transfer closures. At a *macro-scale* level the external void generated by the randomly packed pellet of different shapes are resolved, but the internal structures of the foams are not resolved and instead are modeled by volume averaging techniques [1] using effective closures developed from DNS studies. Studies at the macro-scale level lead to a phenomenological model for packed bed in large industrial applications where the full bed is represented by a single porosity, sometimes referred as the overall volumetric porosity of the bed.

In this present contribution we are focusing on the very detailed pore-scale level studies of open-cell solid foams for single-phase flow and heat transfer analysis. Due to the random and complex geometrical shapes, most of the numerical work on solid foams is based on idealized representations of the foam unit cell. However, the demand for modeling very complex irregular structures has

* Corresponding author.

E-mail address: N.G.Deen@tue.nl (N.G. Deen).

Nomenclature

A_s	Solid surface area, m^2
$\bar{C}$	Convective momentum flux, N/m^3
C_d	Non-dimensional drag coefficient
C_p	Specific heat, $J/(kg \cdot K)$
$\bar{C}_T$	Convective thermal flux, W/m^3
$\bar{d}$	Probe location, m
D	Diameter, m
d_p	Equivalent particle diameter, m
$F_{f \rightarrow s}$	Total force on the solid body, N
f	Friction factor
h	Heat transfer coefficient, $W/(m^2 K)$
k	Thermal conductivity, $W/(m K)$
l	Solid surface to probe distance, m
Nu	Nusselt number
$\bar{n}$	Surface normal
p	Pressure, Pa
Pr	Prandtl number
$\dot{q}$	Wall heat flux, W/m^2
Re	Reynolds number
S_a	Specific surface area, $1/m$
T	Temperature, K
t	Time, s
u	Velocity, m/s
V_s	Solid volume, m^3
$\bar{x}$	Position vector of cell-center, m
$\Delta x, \Delta y$ and Δz	Grid spacing, m
Δt	Time-step, sec

Greek symbols

ρ	Density, Kg/m^3
μ	Viscosity, $Pa \cdot s$
δp	Pressure correction, Pa
ϕ	Any flow variable
ξ	Dimensionless distance
ε	Porosity

Subscripts

c	Center
e	East
IB	Immersed boundary
f	Fluid
S	Surface
s	Solid
sup	Superficial
w	West
∞	Free-stream

increased with the development of X-ray micro computed tomography (μ -CT) imaging [2–4] and available computational resources. This demands the development of closures with a more realistic geometry and decreases the level of uncertainties for the subsequent industrial scale analysis. A 3D image data from a μ -CT scan of an actual foam geometry is usually converted into a surface mesh of unstructured triangular elements and the current article focuses on development of a numerical framework to incorporate such surface mesh as an immersed boundary (IB) in a Cartesian computational domain.

The Immersed Boundary Method [5] is a novel computational technique which is an alternative to the body conformal grid methods for flow over complex shapes. It avoids the challenging high quality grid generation process and the development of a CFD code for unstructured grids. Moreover, simple structured grid based

codes are more efficient than unstructured grid codes in terms of memory requirements to store the grid information, computational costs in terms of convergence characteristics and the efforts involved to generate a good quality mesh. At the same time IBM can easily be coupled with other transport equations [6]. On the other hand, body fitted grids have the great ability for local mesh refinement in zones of interest where sharp gradients occur, whereas for the structured meshes grid refinement would be uniform – increasing the total computational cost. There exists a large variety of IBM methods and interested readers may refer to the literature review by Mittal and Iaccarino [7] on different immersed boundary methods. They classified the existing methods in two categories, namely continuous forcing approaches and sharp-interface based approaches. In the continuous forcing approach the no-slip condition for velocity at the immersed boundary was applied by introducing a continuous 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). The major drawback of such formulations, is the smearing of the boundaries due to the spreading nature of the forcing function. On the other side, the discrete (or direct) forcing approach is more elegant where the IB retains a sharp solid interface with no spatial spreading. The extension of the immersed boundary method from momentum to heat transfer or species transfer is quite natural and straightforward when a solid boundary is subjected to a constant temperature or a constant species concentration (Dirichlet) condition. As compared to the Dirichlet condition, heat transfer problems with the Neumann type (given heat flux) boundary condition are comparatively more challenging. To the best of our knowledge, there exist only a few attempts to properly implement Neumann condition in existing IB techniques.

Several authors like Zhang et al. [23], Kim et al. [24], Pacheco et al. [25] and Pan [26] have implemented both the temperature Dirichlet and Neumann-type boundary conditions using a forcing function based IBM. Tseng and Ferziger [27] used polynomial re-constructing techniques to implement the pressure Neumann condition in their ghost-cell IBM. In that sharp interface method they often found numerical instability and boundedness problems due to large negative coefficients in the discretized Poisson equation. In heat transfer analysis often the energy equation needs to be solved simultaneously in both solid and fluid subdomains, this is usually referred to as conjugate heat transfer (CHT). Marella et al. [9] developed a simple CHT model for solidification using a level-set field. Recently, Nagendra et al. [8] proposed a detailed CHT model using ghost cell IB techniques. They extended a ghost-cell immersed boundary method proposed by Gilmanov and Sotiropoulos [28] for different thermal conditions. The aim of this work is to develop an accurate and efficient sharp interface based IBM to simulate flow and heat transfer through complex porous medium. The present methodology is an extension of the sharp interface, implicit, second order accurate IBM proposed by Deen et al. [5]. The key inclusions in this current contribution are: (a) extension of the method for complex random shaped solid objects, (b) temperature Neumann boundary condition at the solid-fluid interface, and (c) a conjugated heat transfer model.

As discussed earlier, in the chemical process industries solid foams are used as a catalyst support to enhance liquid-solid contacting to facilitate high mass transfer rates. Catalytic reactions are often exothermic or endothermic. Based on the type of reactions and the solid thermal conductivity (i.e. the Biot number), the foam surface exhibits either a constant wall temperature or a constant heat flux condition. For instance, a very fast reaction taking place at the solid surface and/or in the case of very high thermal conductivity of the solid give rise to a uniform temperature inside the solid, i.e. a Dirichlet condition can be assumed for the thermal fluid

Fig. 1. Multi-scale modeling approach for a packed bed filled with porous open-cell solid foam pellets. Subfigure (a), (b) and (c) show the different levels of homogenization or averaging. At the DNS level, the micro-pores are fully resolved by considering a small section of a porous foam sample inside a pellet. At the next level of homogenization the flow inside the individual porous pellets is not resolved, but instead its average behavior is accounted for based on closures developed from DNS studies. In this case, the pores between the pellets are resolved. The results obtained from this kind of averaged model can be used to form phenomenological 1-D models where the full bed is considered to be filled by single homogeneous porous medium.

solid coupling (see e.g. Deen et al. [29]). On the other hand, a slow reaction and/or low solid thermal conductivity create a temperature profile inside the solid. For such cases it is more realistic to use Neumann condition at the solid surface to describe the fluid solid heat transfer. The value of the heat flux at the foam surface depends on the rate of reactions. At the macroscopic level the effective thermal conductivity is also required to simulate heat transfer. To develop closures for the effective thermal conductivity of open-cell foam, we need to solve heat transfer through both the solid and fluid subdomain, i.e. a conjugate heat transfer model is essential [22].

In this paper after discussing the numerical methodology, we present several validation cases using simple geometries. Subsequently, to demonstrate the capability of the current method it is applied to study the flow and heat transfer through a section of a typical complex foam structure generated from a μ -CT scan imaging technique. A pressure drop correlation as a function of Re and a correlation for the foam to fluid heat transfer coefficient is proposed which can be used in more coarse-grained simulation techniques.

2. Model description

2.1. Numerical model and solution methodology

The flow and heat transport in the porous foam is described by the conservation equations of mass, momentum and energy, which for an incompressible, Newtonian fluid read:

$$\nabla \cdot \bar{u} = 0 \quad (1)$$

$$\rho_f \frac{\partial \bar{u}}{\partial t} + \rho_f \nabla \cdot (\bar{u} \bar{u}) = -\nabla p + \mu_f \nabla^2 \bar{u} \quad (2)$$

$$\rho_f C_{pf} \frac{\partial T_f}{\partial t} + \rho_f C_{pf} \bar{u} \cdot \nabla T_f = k_f \nabla^2 T_f \quad (3)$$

where ρ_f , μ_f , C_{pf} and k_f are the fluid density, viscosity, specific heat and conductivity respectively. In the current Finite Volume (FV) implementation transport equations are time-integrated for each staggered computational grid cell. The discretized form of the momentum equation is obtained as:

$$\rho_f \bar{u}^{n+1} = \rho_f \bar{u}^n + \Delta t \left[-\nabla p^{n+1} + \mu_f \nabla^2 \bar{u}^{n+1} - \left\{ \bar{C}_{FOU}^{n+1} + \left(\bar{C}_{Min-Mod}^n - \bar{C}_{FOU}^n \right) \right\} \right] \quad (4)$$

where n indicates the time level and $\bar{C}$ indicates the net-convective flux (momentum):

$$\bar{C} = \rho_f (\nabla \cdot \bar{u} \bar{u}) \quad (5)$$

A deferred correction method has been incorporated in the convection term, where both the First Order Upwind (FOU) scheme and the total variation diminishing (TVD) Min-Mod scheme are used to calculate the convection flux. The flux based on the FOU scheme ($\bar{C}_{FOU}$) is used as a predictor and treated implicitly, while the difference between the TVD and FOU schemes is used as a corrector that is treated in an explicit manner. The second order accurate central difference (CD) scheme is used for discretizing the diffusion terms. The discretized momentum equation is solved with a fractional step method, where at the first-step the tentative velocity field $\bar{u}^{**}$ is computed from:

$$\rho_f \bar{u}^{**} = \rho_f \bar{u}^n + \Delta t \left[-\nabla p^n + \mu_f \nabla^2 \bar{u}^{**} - \left\{ \bar{C}_{FOU}^{**} + \left(\bar{C}_{Min-Mod}^n - \bar{C}_{FOU}^n \right) \right\} \right] \quad (6)$$

To find $\bar{u}^{**}$ we need to solve a set of linear equations. To this end, we use a robust and efficient parallel Block Incomplete Cholesky Conjugate Gradient (B-ICCG) solver. The velocity at the new time level $n+1$ can be obtained from:

$$\bar{u}^{n+1} = \bar{u}^{**} - \frac{\Delta t}{\rho_f} \nabla (\delta p) \quad (7)$$

where $\delta p = p^{n+1} - p^n$, represents the pressure correction. Since $\bar{u}^{n+1}$ need to satisfy the continuity equation, the pressure Poisson equation is obtained as:

$$\nabla \cdot \left\{ \frac{\Delta t}{\rho_f} \nabla (\delta p) \right\} = \nabla \cdot \bar{u}^{**} \quad (8)$$

which is solved by the B-ICCG sparse matrix solver. Similarly, the energy equation is discretized as,

$$C_{pf} \rho_f T_f^{n+1} = C_{pf} \rho_f T_f^n + \Delta t \left[k_f \Delta^2 T_f^{n+1} - \bar{C}_T^n \right] \quad (9)$$

where $\bar{C}_T$ indicates the net thermal convective flux:

$$\bar{C}_T = \rho_f C_{pf} \bar{u} \cdot \nabla T_f \quad (10)$$

The total variation diminishing (TVD) Min-mod scheme is used to calculate the thermal convection flux and the second order accurate central difference (CD) scheme is used for conduction. The algebraic form of Eq. (9) is again solved by the B-ICCG sparse matrix solver. For simulations involving conjugate heat transfer, additionally an energy equation is solved inside the solid structure, which reads:

$$\rho_s C_{ps} \frac{\partial T_s}{\partial t} = k_s \nabla^2 T_s \quad (11)$$

ρ_s , C_{ps} and k_s are the solid density, specific heat and conductivity respectively. Eq. (11) is solved in a similar manner as Eq. (3) with zero velocity field inside the solid.

2.2. Immersed boundary method (IBM)

An implicit (direct) second-order accurate IBM method proposed by Deen et al. [5] has been extended for complex geometries and different thermal boundary conditions at the solid-fluid interface. There exists a large variety of CAD mesh formats to represent complex geometries. In the current framework we have chosen a surface mesh with triangular elements to represent all immersed bodies. Triangular elements are stored in a STL (Stereo-Lithography) file format and are described as a raw unstructured triangulated surface (Fig. 2-a) by the unit normal and vertices (ordered by the right-hand rule) of the triangles using a three-dimensional Cartesian coordinate system. The normal of each triangle always points outwards (i.e. into the fluid). The resolution of the triangular elements of the surface-mesh should be in the order of the background Euler grid spacing for proper representation of the solid body.

Fig. 2-b schematically represents the current IBM implementation. The fluid-solid coupling (i.e. boundary conditions at the fluid-solid interface) is handled at the level of the discretized momentum and energy equations. In this direct or implicit method at first all the cells are marked/flagged as either “solid” (cell-center inside the solid body) or “fluid” (cell-center in the fluid zone). “IB-fluid cells” are special fluid cells that neighbor at least one solid cell and “IB-solid cell” are the special solid cells that are adjacent to at least one fluid cell. In other words, the solid surface passes through between the IB-fluid and IB-solid cells. The wall boundary condition for the fluid region is enforced by special treatment at all IB-fluid cells, whereas for conjugate heat transfer case the wall boundary condition for the solid region is imposed by IB-solid cell. The identifying solid cell inside the surface mesh is a classical ‘point-in-polygon’ problem in 3D. The following steps allow marking the appropriate cells in staggered computational grid:

- (1) At first, the centroid of the each triangular surface elements is located in the Eulerian background grid. A 3D bounding box around the surface element is created and for all the cells in

the bounding box it is checked if they are either solid or fluid-cells. To do so, for each cell in the bounding box, grid lines are drawn connecting the cell center of the cell with each of its six neighboring cell centers. If any of these grid lines intersect the triangular element, the following dot product is evaluated for that pair of cells:

$$v = (\bar{x}_c - \bar{x}_n) \cdot \bar{n} \quad (12)$$

where $\bar{x}_c$ is the position vector of the cell in the bounding box and $\bar{x}_n$ is the position vector of its neighboring cell, $\bar{n}$ is the surface normal of the triangular element. If $v \leq 0$, cell ‘n’ is marked as a solid-cell, else ‘c’ is marked as a solid-cell (more precisely IB-solid cell). In a flag matrix the marked cells are represented as integers: solid-cells get flag –1 and fluid-cells get flag 0. This procedure will create a closed hollow shell of solid flags around the IB. For proper enclosing, the size of the bounding box should be based on the relative size between the triangular element and Eulerian grid. For instance if the size of the triangular element is in the range to the size of the mesh, a stencil size of $3 \times 3 \times 3$ is sufficient.

- (2) IB-fluid cells are marked in a similar way. A cell ‘c’ will be flagged as a IB-fluid cell when one of its neighbor cells (‘n’) is a solid cell and $v \leq 0$ (Eq. (12)). In the flag matrix the IB-fluid cells are initially identified by integer 1 and later changed to a hash code, which is discussed later.
- (3) Until now we have flagged all the IB-solid cells very near to the solid interface (enclosing the solid interface) and all the IB-fluid cells. Now we need to assign solid flag inside the solid body, which were initially flagged as fluid cell. This has to be done by sequential tracing in any of the three directions: for instance if we move along the positive x-direction, then any fluid-cell with index $[i][j][k]$ that have a neighboring solid-cell at index $[i-1][j][k]$ will be marked as a solid-cell. The procedure automatically stops when it reaches a solid cell on the other end of the body as it is guarded by an IB-fluid cell. If the triangular surface mesh is not closed, i.e. a leak or a defect in the surface mesh, the initial marking of the IB-solid and IB-fluid cells will not be closed and as a result the method will fail. For successful representation of the IB in a CFD domain we need a closed, defect free surface mesh, which may be obtained by using CAD software.

The boundary condition for flow on the IB is enforced directly through the IB-fluid cells. The algebraic form of the discretized momentum equation (Eq. (6)) or energy equation (Eq. (9)) at cell-center ‘c’ in Fig. 2-b can be expressed as:

$$a_c \phi_c + \sum_{nb} a_{nb} \phi_{nb} = b_c \quad (13)$$

where ϕ corresponds to one of the fluid velocity components or temperature. ‘nb’ indicates the cell-center of surrounding neighboring cells. The coefficients ‘a’ depend on the fluid properties and the chosen grid resolution. All the explicit terms are contained in b_c . For each IB-fluid cell ‘c’, the fluid-solid boundary condition is imposed by changing the central coefficient a_c and neighboring coefficients a_{nb} of Eq. (13). We will first discuss briefly the implementation of the Dirichlet boundary condition and then its extension to the Neumann boundary condition.

2.2.1. Dirichlet boundary condition

In the Dirichlet boundary condition the value of ϕ at the solid surface is known. In Fig. 2-b IB-fluid cell ‘c’ neighbors a IB-solid cell ‘w’ and a fluid-cell ‘e’. The value of ϕ at the solid surface is ϕ_s . For

Fig. 2. Incorporation of the Dirichlet boundary condition at the surface of an Immersed Body (IB). The IB is represented by a surface mesh of triangular elements (line segment in 2D). (a) The surface mesh for a sphere is made by triangular elements. In the STL file format the vertex of each triangle along with surface normal are stored to generate the solid surface. The surface normal always points outward. (b) The current IBM implementation procedure for irregular shaped bodies is schematically shown in 2D. At the beginning the computation cells are flagged either as 'solid cell', 'fluid cell' or 'IB-fluid cell'. The intersection between the grid line and the IB surface i.e. ξ_s is required to implement the Dirichlet boundary condition and is stored in a Hash Table format. Hash codes are stored in the flag matrix for IB-solid cells and each of these point to a Table where six ξ_s values are stored.

Fig. 3. The implementation of (a) constant heat flux (CHF) case and (b) conjugate heat transfer (CHT) case. For CHF the temperature at the intersection between the grid line with solid surface ($\bar{d}_s$) is evaluated by creating two probe points ($\bar{d}_1$ & $\bar{d}_2$) in the direction normal to the surface. Whereas for the case CHT it requires total four probes, two inside the solid and two in the fluid region. The temperature at the probe locations is evaluated by trilinear interpolation based on the old time-step temperature field. Once the wall temperature at $\bar{d}_s$ is calculated, we store it in the hash Table and use the Dirichlet condition implementation for the fluid-solid thermal coupling.

momentum equation $\phi_s = 0$, i.e. the no-slip boundary condition at the solid wall and for the energy equation it is $\phi_s = T_{wall}$, i.e. a constant wall temperature. To enforce solid-fluid coupling the generalized flow quantity ϕ in the IB-solid cell 'w' is expressed as a linear combination of relevant fluid-cells in such a way that it satisfies $\phi = \phi_s$ at the wall ('s'). For this purpose a quadratic fit is used: $\phi = p\xi^2 + q\xi + r$, where ξ is the dimensionless distance from

IB-solid cell 'w' as shown in Fig. 2-b. The values of coefficients p , q and r are obtained from the known values of ϕ_s at the IB, which leads to:

$$\phi_w = -\frac{2\xi_s}{1-\xi_s}\phi_c + \frac{\xi_s}{2-\xi_s}\phi_e + \frac{2}{(1-\xi_s)(2-\xi_s)}\phi_s \quad (14)$$

Fig. 4. The non-dimensional drag force (C_D) with Re for flow past a circular cylinder placed in a uniform velocity field. A very good agreement is found between the present result and existing experimental and numerical data for a wide range of Re .

The modified coefficients ($\hat{a}$) for Eq. (13) came out as:

$$\hat{a}_c = a_c + a_w \left(-\frac{2\xi_S}{1 - \xi_S} \right) \quad (15a)$$

$$\hat{a}_e = a_e + a_w \left(-\frac{\xi_S}{2 - \xi_S} \right) \quad (15b)$$

$$\hat{b}_c = b_c - a_w \left(\frac{2}{(1 - \xi_S)(2 - \xi_S)} \right) \quad (15c)$$

$$\hat{a}_w = 0 \quad (15d)$$

Note that Eq. (13) is also formed for solid-cells (and IB-solid cells) and to enforce zero velocity and/or a constant temperature condition the central coefficient (a_c) is set to unity, while all neighboring coefficients (i.e. a_{nb}) are set to 0 and $b_c = \phi_S$.

From the above methodology proposed by Deen et al. [5] it is clear that to accommodate the IB in the Cartesian grid we only need the ξ_S values at the IB-fluid cells. It can easily be calculated by the intersection between the triangular element and the grid line(s) as shown in Fig. 2-b. For a stationary IB, to reduce the computational time drastically, we store ξ_S for each IB-fluid cells in a hash Table format (Fig. 2-b), which is a data structure where a key maps to the associated values. As shown in Fig. 2-b all the IB-fluid cells are numbered and these numbers are stored in a flag matrix. These numbers represent a hash code or key for the hash Table where all six (for 3D) ξ_S values are stored. Please note that if an IB-fluid cell shares only one neighboring solid-cell (which happens for most of the IB-fluid cells) only one value of ξ_S is needed and the rest are irrelevant (stored as 0 in the code). Once the hash Table is generated, the triangular surface mesh is no longer needed for any stationary body case.

2.2.2. Neumann boundary condition

In this case instead of prescribing the exact value of ϕ at the surface the gradient of ϕ normal to the solid wall is given. For constant heat flux case at the solid boundary we need to specify,

$$-k_f \frac{\partial T_f}{\partial \bar{n}} \Big|_{\text{wall}} = \dot{q}_{\text{wall}} \quad (16)$$

where $\dot{q}_{\text{wall}}$ is the constant prescribed wall heat flux, k_f the fluid conductivity and $\bar{n}$ the surface normal pointing outward of the solid surface.

The most appropriate choice for specifying the Neumann condition is to use Eq. (16) to compute the temperature at the solid surface and then use the Dirichlet boundary condition framework presented before. Note that we use the temperature field at the old-time level to compute the “fixed” wall temperature for the

Table 1

Computed Nu values for flow past a static cylinder with constant heat flux condition at the cylinder wall ($Pr=0.7$). Unless specified otherwise all reported data are obtained with the G20 grid.

Re	Present IBM	Bharti et al. [15]	Nagendra et al. [8]	Ren et al. [16]
10	2.03	2.04	2.00	2.03
20	2.76	2.78	2.71	2.74
40	3.75	3.77	3.67	3.74
100	G10: 6.468 G20: 5.896 G40: 5.873	–	–	5.886
150	7.15	–	–	–
200	8.09	–	–	–

new-time level. Though in this way the wall temperature lags one time-step, the implicit implementation of the Dirichlet condition maintains the stability of the solution. Similar to ξ_S the wall temperature at the wall-grid line intersection point are stored in the hash Table but this time they are updated at each time-step. In Fig. 3-a the detailed implementation is presented. To compute the surface temperature at the surface-grid line intersection point, i.e. at $\bar{d}_S$, probes normal to the surface are created. For second order accuracy interpolation to calculate surface temperature, two probes are created towards the fluid-cells normal to IB. The probes are placed at a distance l and $2l$ from the solid interface:

$$\bar{d}_1 = \bar{d}_S + l \cdot \bar{n}, \quad \bar{d}_2 = \bar{d}_S + 2l \cdot \bar{n} \quad (17)$$

The temperature at the probe location is calculated by trilinear interpolation using the temperature field from the previous time level. It is a natural choice to use $l = \Delta x$, where Δx is the uniform Eulerian grid spacing. After the temperature field is interpolated to the probe locations, the surface temperature is calculated by a Taylor series expansion of Eq. (16), giving:

$$T_S = \frac{1}{3} (4T_1 - T_2 - 2Gl) \quad (18)$$

where T_1 and T_2 are the interpolated temperature at the probe $\bar{d}_1$ and $\bar{d}_2$. For constant heat flux case:

$$G = -\dot{q}_S / k_f$$

For certain solid interface orientation the second probe may lie in the solid or even outside the CFD domain. For such cases, a first order formulation is used instead, which involves only a single probe normal to the interface. From Eq. (16) the surface temperature is calculated as,

$$T_S = T_1 - Gl \quad (19)$$

Fig. 5. The surface averaged Nu number for different Re numbers for flow past a circular cylinder cooled by a free fluid stream ($Pr=0.7148$). At the cylinder wall a constant temperature is specified. A very good agreement is found between the present results and the o-grid based numerical results by Lange et al. [14] for a wide range of Re .

Fig. 6. Conjugate heat transfer analysis for a solid sphere placed in a stagnant fluid. The sphere is represented by a triangulated surface mesh. The conductivity ratio between the solid and fluid phases is 350. At the z_{min} wall $T=100\text{ K}$ and at the z_{max} wall $T=600\text{ K}$ is specified. At steady state, proper implementation of the solid-fluid coupling should provide zero heat flux across the closed surface. In the present study we find a negligible heat flux through the IB surface ($\sim 10^{-9}\text{ W/m}^2$).

2.2.3. Conjugate heat transfer case

For the cases of conjugate heat transfer study the energy equation need to be solved in the fluid region as well as the solid region. Solid material properties are assigned for the cells whose cell-centers lie inside the solid region. Along with this for the present sharp interface IBM we need to enforce continuity of the temperature and normal heat flux at the solid-fluid interface, i.e.

$$-k_f \frac{\partial T_f}{\partial \bar{n}} \Big|_s = -k_s \frac{\partial T_s}{\partial \bar{n}} \Big|_s \text{ and } T_f|_s = T_s|_s \quad (20)$$

Sub-scripts 's' and 'f' represent solid and fluid respectively. Similar to the Neumann condition implementation, the idea is to find the solid surface temperature to satisfy Eq. (20) and then use it as Dirichlet boundary condition. At the beginning of each time step, the temperature at the intersection between the solid surface and the grid line is calculated and stored it in the hash Table format. Total four probes are used – two in the fluid and two in the solid direction – to calculate solid surface temperature from Eq. (20). For 2-D case it is schematically shown in Fig. 3-b.

$$\bar{d}_1 = \bar{d}_s + l \cdot \bar{n}, \quad \bar{d}_2 = \bar{d}_s + 2l \cdot \bar{n}, \quad \bar{d}_3 = \bar{d}_s - l \cdot \bar{n}, \quad \bar{d}_4 = \bar{d}_s - 2l \cdot \bar{n} \quad (21)$$

By Taylor series expansion of Eq. (20) the wall temperature is calculated as:

$$T_s = \frac{k_f(4T_1 - T_2) + k_s(4T_3 - T_4)}{3(k_f + k_s)} \quad (22)$$

T_1 and T_2 are the temperatures at the probes ($\bar{d}_1$ and $\bar{d}_2$) on the fluid side and T_3 and T_4 are the solid side probe ($\bar{d}_3$ and $\bar{d}_4$) temperatures. In Fig. 3-b F_1 and F_2 are the two fluid cells and S_1 and S_2 are the two solid cells. While solving for F_1 , the coefficients of the discretized energy equation are modified based on Eq. (15). Along with solid surface temperature (T_s) it requires the non-dimensional distance (along the grid-line) from IB-solid cell (S_1) to the interface (i.e. ξ_s), which is already available in the hash Table of the cell F_1 . Similarly for the solid cell S_1 the coefficients of the energy equation are modified using the solid surface temperature (T_s) and the non-dimensional distance from IB-fluid cell (F_1) to the interface (i.e. $1 - \xi_s$). As a result the calculation of the temperature fields in the fluid and the solid cells are decoupled and solved with their individual wall boundary conditions. For very complex structures it may happen that the second fluid probe $\bar{d}_2$ (second solid probe $\bar{d}_4$) lies in the solid (fluid) or outside the CFD domain. For such cases only linear interpolation is feasible to calculate the solid surface temperature according to:

$$T_s = \frac{T_1 k_f + T_3 k_s}{k_f + k_s} \quad (23)$$

2.2.4. Discussions

It is worth mentioning that, when ξ_s is very close to unity i.e. IB-fluid cell ('c' in Fig. 2-b) is very near to a solid surface, some terms in Eq. (14) and 15 will be singular. To overcome this, we set a criteria such that for $\xi_s > 0.99$, 'c' is assigned as a solid-cell and 'e' to IB-fluid cell with $\xi_s = 0$ [21]. For a very complex geometrical shaped porous structure, it may happen that two triangular elements representing the solid surface are very close to each other and as a result for few IB-fluid cells the quadratic interpolation is not possible [21]. For such cases we use a linear fit (instead of a quadratic fit) to change the coefficients. As the number of such cells are very less, the overall accuracy of the solution remains almost the same.

For the constant heat flux and CHT cases it is interesting to study the effect of the probe distance from solid surface (i.e. l) on the overall accuracy. For low and moderate Re case we have tested the accuracy of the solution by choosing $l=0.7\Delta x$ and $l=1.3\Delta x$, and we found only very small deviations between the two cases ($\sim 1\%$) on the temperature variation and overall heat transfer coefficient. However for higher Re cases, a very thin boundary layer forms, and there is a potential risk that for a specific grid resolution the second probe lies outside the boundary layer. As a result it significantly reduces the accuracy of the solution. For such cases it is suggested to calibrate the probe location or apply further grid refinement.

For constant heat flux and CHT cases, at a particular probe location, the temperature value is evaluated by trilinear interpolation which involves volumetric averaging of the temperature using the eight grid cells surrounding the probe. It may happen, depending upon the solid geometry, that some of the surrounding cell-centers lie inside the solid. For conjugate heat transfer, the temperatures of the solid cells are also solved, however for cases of a constant heat flux this requires to evaluate the solid temperature at the IB-solid cells. The temperature at all the IB-solid cells is calculated by quadratic extrapolation of the fluid region temperature satisfying the Neumann (Eq. (16)) at the solid wall. Another solution to this issue would be to exclude solid cells from the volumetric averaging during trilinear interpolation. This was not attempted in this work though.

In the current method we do not need to perform any strict IB treatment for the pressure field as the velocity field inside the solid

Table 2
Simulation settings for flow past a section of a typical solid foam sample.

Parameters	Values	Unit
Computational cell	$552 \times 230 \times 230$	(–)
$\Delta x = \Delta y = \Delta z$	4×10^{-5}	m
Time step (Δt)	10^{-5}	s
ρ_f	1.1, 110.8, 221.6, 554.1, 1108.2	kg/m ³
μ_f	0.01	kg/(m s)
k_f	1	W/(m K)
C_{pf}	100	J/(kg K)
u_{sup}	1	m/s
T_{inlet}	300	K
T_{wall}	400	K

body is already divergence free, which explicitly satisfies $\partial p / \partial \bar{n} = 0$ at the solid surface. The pressure field is also computed inside the solid body and the pressure correction step (Eq. (8)) leads to a very small value of the velocity ($O(u_{sup} \times 10^{-3})$) inside the solid body, which has an insignificant effect on the overall simulation results. However, implicit implementation of the pressure Neumann condition at the solid would solve this issue, which is not attempted in the present case.

3. Verification and validation

The framework described in the previous section has been rigorously verified and validated module by module for a large range of problems and test cases. For all cases the geometry is represented by a triangulated surface mesh and we found very good agreements with literature results. In this paper we only present two standard test cases to show the accuracy of the method. These cases involve the flow past a circular cylinder and the conjugate heat exchange between a stagnant fluid and a solid sphere.

3.1. Flow past a circular cylinder

The flow around a circular cylinder placed in infinite domain has become the standard for testing the accuracy of many computational techniques due to the availability of a wide variety of experimental and numerical results. Here we have performed flow and heat transfer simulation up to $Re = 200$ (in that case the flow remains approximately 2D). Both constant wall temperature (CWT) and constant heat flux boundary (CHF) conditions were tested at the cylinder wall. For different cases the drag coefficient (C_d) and the surface averaged Nusselt number (Nu) have been compared, which are defined as:

$$C_d = \frac{F_{f \rightarrow s}}{(1/2)\rho_f U_\infty^2} \quad (24)$$

$$Nu_{CWT} = \frac{D}{k_f} \frac{1}{(T_{wall} - T_\infty)A_s} \int_{A_s} \dot{q}_{wall,loc} dA_s \quad (25)$$

$$Nu_{CHF} = \frac{D}{k_f} \frac{\dot{q}_{wall}}{A_s} \int_{A_s} \frac{1}{(T_{s,loc} - T_\infty)} dA_s \quad (26)$$

where $F_{f \rightarrow s}$ is the total force on the solid body (this includes the force due to viscous shear and the pressure force), D is the diameter of the cylinder, A_s is surface area of the cylinder. U_∞ and T_∞ are the free stream velocity and temperature. For the CWT case $\dot{q}_{wall,loc}$ is the (variable) wall heat flux from the cylinder to fluid, calculated from the temperature gradient at the cylinder surface, whereas T_{wall} is the constant wall temperature specified at the cylinder wall. For the CWF case $\dot{q}_{wall}$ is the constant wall heat flux constant at the cylinder wall and $T_{s,loc}$ is the (variable) local cylinder wall temperature calculated based on Eq. (18).

A pseudo-2D rectangular computational domain is considered where only three grid cells are placed along the axial direction of

the cylinder. The domain size in the flow direction is $45D$ and in the lateral direction it is $30D$. The cylinder is placed at a distance $20D$ from the inlet and in the middle between the other two boundaries. A uniform plug flow velocity (constant temperature) at the inlet, a pressure outflow (zero temperature gradient) boundary condition at the outlet and free-slip (adiabatic wall) boundary conditions for the other walls have been specified for the flow (energy) solver.

For the CWT case the Prandtl number (Pr) is set equal to 0.7148 and for the CHF case $Pr = 0.7$ is chosen to enable one-to-one comparison with available literature data. For all the cases the diameter of the cylinder is resolved by 20 grid cells. The variation of C_d with Re is shown in Fig. 4 along with existing numerical and experimental results [10–13]. In Fig. 5 the variation of the average Nu number with Re for the CWT case is compared with o-grid (a structured polar grid wrapping the cylinder) based numerical results by Lange et al. [14]. For the CHF case the variation of the average Nu number with Re is tabulated in Table 1. A very good agreement is observed. To check the effect of the grid resolution, a constant heat flux case was performed for $Re = 100$, using three different grids, G10: 10 grid cells per cylinder diameter, G20: 20 grid cells per diameter and G40: 40 grid cells per diameter. It is found that the calculated Nu using G2 provides grid independent results.

3.2. Conjugate heat exchange between stagnant fluid and a solid sphere

To test the implementation of the conjugate heat transfer module we present a simple test case similar to that described by Kedia et al. [17] where a solid object is placed in a stagnant fluid zone. In our case a surface mesh of a solid sphere is placed in the center of the cubic fluid zone. As the fluid is stagnant, only the energy equation is solved and at the z_{min} boundary $T = 100$ K and at the z_{max} boundary $T = 600$ K conditions are specified. All the other four walls are set to adiabatic conditions. The conductivity ratio between the solid and the fluid is chosen as 350 ($k_f = 1$ W/m-K and $k_s = 350$ W/m-K). A similar conductivity ratio is found in a water-aluminum system. Proper implementation of the solid-fluid coupling should guarantee that the heat-flux in from the z_{max} wall equals the heat-flux out from the z_{min} wall as well as a zero total heat flux from the closed solid body ($\dot{q}_{IB}$) at steady state. The size of the fluid domain is $160 \text{ mm} \times 160 \text{ mm} \times 160 \text{ mm}$ and diameter of the solid sphere is 80 mm . Grid size of $40 \times 40 \times 40$ has been chosen (i.e. 20 grid cells across the sphere). In Fig. 6 the steady state simulation results are shown. We found a negligible total heat flux from the closed IB surface, which indicates the accuracy of the method. The porosity of the system is 0.9345 and the stagnant effective conductivity is found as 7.55 W/m-K.

4. Flow through a typical random solid foam structure

After developing and validating the numerical code the attention has been turned to applying the model for the simulation of the flow past a typical random foam structure. The main aim here is to develop a closure for momentum and energy transport for a specific foam sample. In Fig. 7 the surface mesh for a typical section of the foam is shown. It is created using 3D image data generated from a μ -CT scan of an actual foam geometry [18]. The section of the foam is chosen such that the main morphological properties like porosity ($\varepsilon = 0.925$) and the specific surface area ($S_a = 6650 \text{ m}^{-1}$) remain within $\pm 0.5\%$ of the original larger sample. The average pore diameter of the foam is $\sim 4 \text{ mm}$. The section of the foam sample in the rectangular CFD domain is shown in Fig. 7. To reduce boundary effects in the simulations, inlet and outlet sections are specified around the foam. A constant plug-flow velocity and constant temperature are defined at the inlet, whereas zero pressure value, and

Fig. 7. A typical open-cell solid foam inside the applied computational domain. The STL surface mesh of the foam is incorporated in the Cartesian computational domain by the immersed boundary method. At the inlet a constant plug flow velocity is specified and the outlet is specified as a pressure outlet. Free-slip boundary conditions are chosen for all the other four walls. Similarly for the energy equation, a constant inlet temperature, constant wall temperature at the IB wall and adiabatic condition for the other four walls are chosen.

Fig. 8. Simulation snapshot of flow through a random foam sample for (left) $Re = 0.1$ and (right) $Re = 100$. For the low Re case the effect of flow velocity (convection) is small, hence due to conduction the fluid reaches almost the same temperature as the foam. On the other hand for the high Re case thermal boundary layers form at the solid walls and substantial temperature differences can be noticed between the average solid and fluid temperature. In the velocity contours of high Re we can observe the vigorous motion of fluid flow, which increases the mixing and enhances the heat transfer rate.

zero gradient conditions for the temperature and velocity are specified at the outlet. All the four side walls are subject to free-slip conditions for the momentum equations and adiabatic conditions for the energy equation. Each solid ligament of the foam is resolved approximately by 15–20 grid cells, which is sufficient to get grid independent results. For the heat transfer simulation a constant wall temperature at the solid foam wall is specified. Starting from creeping flow Re is varied up to 100. Re is calculated based on the

superficial velocity (u_{sup}) and the characteristic length scale, which is chosen based on the equivalent spherical diameter as,

$$d_p = \frac{6V_s}{A_s} \quad (27)$$

where V_s and A_s are the solid volume and surface area in the domain respectively. The equivalent diameter of this foam sample is calculated as 0.9024 mm . Pr of the fluid is chosen as 1. To obtain the

Fig. 9. Non-dimensional drag force (f) with Re for the present foam sample. The present simulation results are well within the correlation by Dietrich [19] derived from several experimental data. However the Ergun correlation underpredicts the pressure drop for foams.

pressure drop correlation we start from the ansatz form of pressure drop:

$$\frac{\Delta p}{\Delta x} = a \frac{\mu}{d_p^2} u_{sup} + b \frac{\rho}{d_p} u_{sup}^2 \quad (28)$$

where actually $a = g(\varepsilon)$ and $b = h(\varepsilon)$. Note that since we are using a single foam sample with a single value of ε , for our case a and b are constant. The pressure drop will be normalized using viscous scale, and as a result the friction factor f , in our case is defined as:

$$f = \frac{\Delta p / \Delta x}{\mu u_{sup} / d_p^2} = a + b Re \quad (29)$$

This form corresponds to the non-dimensional form of the Ergun equation for packed beds:

$$f_{Ergun} = \frac{150(1-\varepsilon)^2}{\varepsilon^3} + \frac{1.75(1-\varepsilon)}{\varepsilon^3} Re \quad (30)$$

Similarly for non-dimensional heat transfer coefficient, i.e. Nusselt number, we use:

$$Nu = \frac{h d_p}{k_f} = p + q Re^r Pr^{1/3} \quad (31)$$

where p , q and r are constants, that need to be evaluated from the DNS results. The heat transfer coefficient (h) in the present case is defined as:

$$h = \frac{1}{(T_{wall} - T_{bulk}) A_s} \int_{A_s} \dot{q}_{wall, loc} dA_s \quad (32)$$

where $\dot{Q}$ total heat flux from the solid surface, T_{wall} is the constant wall temperature, and T_{bulk} is the inlet bulk temperature. The simulation settings are described in Table 2.

In Fig. 8 the simulation results are shown for $Re = 0.1$ and $Re = 100$. For the case of low Re the effect of the flow velocity or convection is small, hence due to conduction the fluid reaches almost the temperature as the foam. On the other hand at high Re thermal boundary layers form at the solid walls and substantial temperature variations can be noticed. From the velocity contours at high Re we can observe the vigorous motion of fluid flow, which increases the mixing and as a result increases the heat transfer rate.

In Fig. 9 the numerically obtained friction factor (f) is plotted as a function of Re . By curve fitting for this specific foam sample ($\varepsilon = 0.925$) we found that $a = 3.0493$ and $b = 0.1828$. Similarly using Eq. (30) one can obtain $a_{Ergun} = 1.066$ and $b_{Ergun} = 0.1658$, which

Table 3

Comparison of different a and b values of pressure drop correlations.

Correlations	a	b
Ergun Correlation	1.066	0.166
Dietrich et al. [19]	1.759	0.206
Present simulation	3.049	0.183

Fig. 10. The non-dimensional heat transfer coefficient (Nu) with Re for the present foam sample. The solid wall of the foam is subjected to a constant temperature boundary condition.

indicates that in this range of Re the Ergun correlation underpredicts the pressure drop particularly in the creeping flow limit, which is characterized by a . The current simulations results are also compared with existing experimental correlation. Dietrich et al. [19] reviewed experimental pressure drop correlation/values from about 20 authors including around 100 different foam samples and proposed a pressure drop correlation. It is based on over 2500 experimental values and most of the data point lies within an error range of $\pm 40\%$ of the proposed pressure drop correlation. In the current non-dimensional form of representation it reads:

$$f_{Dietrich} = \frac{247.5(1-\varepsilon)^2}{\varepsilon^3} + \frac{2.175(1-\varepsilon)}{\varepsilon^3} Re \quad (33)$$

Hence, $a_{Dietrich} = 1.759$ and $b_{Dietrich} = 0.2061$. Fig. 9 shows that for moderate Re the present simulation results lie well within the error range of the experimental correlation. In Table 3 the different values of a and b are tabulated.

The same simulations were used to obtain a heat transfer closure. Fig. 10 shows the variation of Nu with Re . Apart from the flow velocity and fluid properties the value of Nu also depends on the solid surface boundary condition (i.e. CWT or CHF), the definition of heat transfer coefficient and the choice of the characteristic length scale. For the current definition, curve-fitting equation proposed as:

$$Nu = 0.2 + 0.1583 Re^{0.7796} Pr^{1/3} \quad (34)$$

5. Conclusions

A sharp implicit Immersed Boundary Method (IBM) for complex geometries is developed to study the hydrodynamics and heat transfer for a random open-cell solid foam. The method provides a versatile and accurate framework for flow through complex geometries, where the presence of solid bodies immersed in the fluid can be incorporated by means of a triangulated surface mesh on a Eulerian Cartesian background mesh. Surface meshes for ordered bodies

are created and used to check the accuracy of the current simulation framework. The simulation results match well with existing literature data. Different thermal boundary conditions at the fluid-solid interface have been incorporated and tested.

Finally, to demonstrate the capability of the current IBM based CFD model, it is used to simulate flow through an irregular solid foam structure. The pressure drop and average Nu as a function of Re were derived for this typical foam structure. The method can easily be extended to study systems with heterogeneous catalytic reactions at the solid-fluid interface, where instead of (or in addition to) a thermal energy equation a mass transport equation needs to be solved with similar boundary conditions [20]. The internal pore structure of the solid foam significantly influences its hydrodynamic behavior and the present method is very helpful for accurate prediction of pressure drop for a specific foam sample.

Acknowledgement

We thank SURFsara (www.surfsara.nl) for the support in using the Cartesius supercomputer.

References

- [1] S. Whitaker, Diffusion and dispersion in porous media, *AIChE J.* 13.3 (1967) 420–427.
- [2] P. Ranut, E. Nobile, L. Mancini, High resolution microtomography-based CFD simulation of flow and heat transfer in aluminum metal foams, *Appl. Therm. Eng.* 69.1 (2014) 230–240.
- [3] E. Bianchi, G. Groppi, W. Schwieger, E. Tronconi, H. Freund, Numerical simulation of heat transfer in the near-wall region of tubular reactors packed with metal open-cell foams, *Chem. Eng. J.* 264 (2015) 268–279.
- [4] E. Bianchi, T. Heidig, C.G. Visconti, G. Groppi, H. Freund, E. Tronconi, Heat transfer properties of metal foam supports for structured catalysts: wall heat transfer coefficient, *Catal. Today* 216 (2013) 121–134.
- [5] N.G. Deen, S.H. Kriebitzsch, M.A. van der Hoef, J.A.M. Kuipers, Direct numerical simulation of flow and heat transfer in dense fluid-particle systems, *Chem. Eng. Sci.* 81 (2012) 329–344.
- [6] N.G. Deen, E.A.J.F. Peters, J.T. Padding, J.A.M. Kuipers, Review of direct numerical simulation of fluid-particle mass, momentum and heat transfer in dense gas-solid flows, *Chem. Eng. Sci.* 116 (2014) 710–724.
- [7] R. Mittal, G. Iaccarino, Immersed boundary methods, *Annu. Rev. Fluid Mech.* 37 (2005) 239–261.
- [8] K. Nagendra, D.K. Tafti, K. Viswanath, A new approach for conjugate heat transfer problems using immersed boundary method for curvilinear grid based solvers, *J. Comput. Phys.* 267 (2014) 225–246.
- [9] S. Marella, S. Krishnan, H. Liu, H.S. Udaykumar, A Sharp interface Cartesian grid method I: an easily implemented technique for 3D moving boundary computations, *J. Comput. Phys.* 210 (1) (2005) 1–31.
- [10] B.N. Rajani, A. Kandasamy, S. Majumdar, Numerical simulation of laminar flow past a circular cylinder, *Appl. Math. Model.* 33 (3) (2009) 1228–1247.
- [11] H. Takami, H.B. Keller, Steady two-dimensional viscous flow of an incompressible fluid past a circular cylinder, *Phys. Fluids* 12 (II-51) (1969) 1958–1988.
- [12] A.L.F.L.E. Silva, A. Silveira-Neto, J.J.R. Damasceno, Numerical simulation of two-dimensional flows over a circular cylinder using the immersed boundary method, *J. Comput. Phys.* 189 (2) (2003) 351–370.
- [13] D. Tritton, Experiments on the flow past a circular cylinder at low Reynolds numbers, *J. Fluid Mech.* 6 (04) (1959) 547–567.
- [14] C.F. Lange, F. Durst, M. Breuer, Momentum and heat transfer from cylinders in laminar crossflow at $10^{-4} \leq Re \leq 200$, *Int. J. Heat Mass Transf.* 41 (22) (1998) 3409–3430.
- [15] R.P. Bharti, R.P. Chhabra, V. Eswaran, A numerical study of the steady forced convection heat transfer from an unconfined circular cylinder, *Heat Mass Transf.* 43 (7) (2007) 639–648.
- [16] W. Ren, S. Chang, Y. Wenming, An efficient immersed boundary method for thermal flow problems with heat flux boundary conditions, *Int. J. Heat Mass Transf.* 64 (2013) 694–705.
- [17] K.S. Kedia, C. Safta, J. Ray, H.N. Najm, A second-order coupled immersed boundary-SAMR construction for chemically reacting flow over a heat-conducting Cartesian grid-conforming solid, *J. Comput. Phys.* 272 (2014) 408–428.
- [18] <https://grabcad.com/library/auxetic-foam-sample-1;>
- [19] B. Dietrich, Pressure drop correlation for ceramic and metal sponges, *Chem. Eng. Sci.* 74 (2012) 192–199.
- [20] N.G. Deen, E.A.J.F. Peters, Johan T. Padding, J.A.M. Kuipers, Review of direct numerical simulation of fluid-particle mass momentum and heat transfer in dense gas-solid flows, *Chem. Eng. Sci.* 116 (2014) 710–724.
- [21] S. Das, N.G. Deen, J.A.M. Kuipers, Immersed boundary method (IBM) based direct numerical simulation of idealized open-cell solid foams: hydrodynamics, *AIChE J.* (2016).
- [22] A. Nakayama, F. Kuwahara, Y. Kodama, An equation for thermal dispersion flux transport and its mathematical modelling for heat and fluid flow in a porous medium, *J. Fluid Mech.* 563 (2006) 81–96.
- [23] N. Zhang, Z.C. Zheng, S. Eckels, Study of heat-transfer on the surface of a circular cylinder in flow using an immersed-boundary method, *Int. J. Heat Fluid Flow* 29.6 (2008) 1558–1566.
- [24] J. Kim, and Haecheon Choi An immersed-boundary finite-volume method for simulation of heat transfer in complex geometries, *KSME Int. J.* 18.6 (2004) 1026–1035.
- [25] J.R. Pacheco, A. Pacheco-Vega, T. Rodić, R.E. Peck, Numerical simulations of heat transfer and fluid flow problems using an immersed-boundary finite-volume method on nonstaggered grids, *Numer. Heat Transf. B-Fund.* 48.1 (2005) 1–24.
- [26] D. Pan, An immersed boundary method on unstructured Cartesian meshes for incompressible flows with heat transfer, *Numer. Heat Tr. B-Fundam.* 49.3 (2006) 277–297.
- [27] Yu-Heng Tseng, J.H. Ferziger, A ghost-cell immersed boundary method for flow in complex geometry, *J. Comput. Phys.* 192.2 (2003) 593–623.
- [28] A. Gilmanov, F. Sotiropoulos, A hybrid Cartesian/immersed boundary method for simulating flows with 3D geometrically complex, moving bodies, *J. Comput. Phys.* 207.2 (2005) 457–492.
- [29] N.G. Deen, J.A.M. Kuipers, Direct numerical simulation of fluid flow accompanied by coupled mass and heat transfer in dense fluid-particle systems, *Chem. Eng. Sci.* 116 (2014) 645–656.