NRC Publications Archive Archives des publications du CNRC

Solution of flow around complex-shaped surfaces by an immersed boundary-body conformal enrichment method

Ilinca, F.; Héту, J.-F.

This publication could be one of several versions: author's original, accepted manuscript or the publisher's version. / La version de cette publication peut être l'une des suivantes : la version prépublication de l'auteur, la version acceptée du manuscrit ou la version de l'éditeur.

For the publisher's version, please access the DOI link below. / Pour consulter la version de l'éditeur, utilisez le lien DOI ci-dessous.

Publisher's version / Version de l'éditeur:

<https://doi.org/10.1002/fld.2615>

International Journal for Numerical Methods In Fluids, 69, 4, pp. 824-841, 2011-06-22

NRC Publications Record / Notice d'Archives des publications de CNRC:

<https://nrc-publications.canada.ca/eng/view/object/?id=382b0072-21e0-4809-8aca-cf2b3ddce654>

<https://publications-cnrc.canada.ca/fra/voir/objet/?id=382b0072-21e0-4809-8aca-cf2b3ddce654>

Access and use of this website and the material on it are subject to the Terms and Conditions set forth at

<https://nrc-publications.canada.ca/eng/copyright>

READ THESE TERMS AND CONDITIONS CAREFULLY BEFORE USING THIS WEBSITE.

L'accès à ce site Web et l'utilisation de son contenu sont assujettis aux conditions présentées dans le site

<https://publications-cnrc.canada.ca/fra/droits>

LISEZ CES CONDITIONS ATTENTIVEMENT AVANT D'UTILISER CE SITE WEB.

Questions? Contact the NRC Publications Archive team at

PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca. If you wish to email the authors directly, please see the first page of the publication for their contact information.

Vous avez des questions? Nous pouvons vous aider. Pour communiquer directement avec un auteur, consultez la première page de la revue dans laquelle son article a été publié afin de trouver ses coordonnées. Si vous n'arrivez pas à les repérer, communiquez avec nous à PublicationsArchive-ArchivesPublications@nrc-cnrc.gc.ca.

Solution of flow around complex-shaped surfaces by an immersed boundary-body conformal enrichment method

F. Ilinca^{*,†} and J.-F. Hétu[‡]

National Research Council, 75 boul de Mortagne, Boucherville, QC, J4B 6Y4, Canada

SUMMARY

Solving the flow around objects with complex shapes may involve extensive meshing work that has to be repeated each time a change in the geometry is needed. Time consuming meshing can be avoided when the solution algorithm can tackle grids that do not fit the shape of immersed objects. This work presents applications of a recently proposed immersed boundary—body conformal enrichment method to the solution of the flow around complex shaped surfaces such as those of a metallic foam matrix. The method produces solutions of the flow satisfying accurately Dirichlet boundary conditions imposed on the immersed fluid/solid interface. The boundary of immersed objects is defined using a level-set function, and the finite element discretization of interface elements is enriched with additional degrees of freedom, which are eliminated at element level. The method is first validated in the case of flow problems for which reference solutions on body-conformal grids can be obtained: flow around an array of spheres and flow around periodic arrays of cylinders. Then, solutions are shown for the more complex flow inside a metallic foam matrix. A multiscale approach combining the solution at the pore level by the immersed boundary method and the macro-scale solution with simulated permeability is used to solve actual experimental configurations. The computed pressure drop as a function of the flow rate on the macro scale configuration replicating two experimental setups is compared with the experimental data for various foam thicknesses. Copyright © 2011 National Research Council Canada

Received 8 February 2011; Revised 4 April 2011; Accepted 18 April 2011

KEY WORDS: immersed boundary method; body conformal enrichment; finite elements; metallic foam; multiscale modeling; permeability

1. INTRODUCTION

Most CFD and fluid-structure interaction solvers are based on body-conformal (BC) grids (i.e., the external boundary and surfaces of immersed bodies are represented by the mesh faces), but there is an increased interest in solution algorithms for non body-conformal grids. For these methods, the spatial discretization is performed over a single domain containing both fluid and solid regions and where mesh points are not necessarily located on the fluid-solid interface [1–3]. For simplicity, we will use in the present work the immersed boundary (IB) term to identify methods using non body-conformal meshes. The IB methods have the main advantage of avoiding costly and sometimes very difficult meshing work on body-fitted geometries. Generally a regular parallelepiped is meshed with a uniform grid. The IB method results also in important algorithmic simplifications when immersed moving bodies are considered. One important drawback of such a method is that the boundary, which has an important influence on the solution especially for fluid-solid interaction, is also a place where distorted elements may be found once regular mesh elements are cut by the

*Correspondence to: F. Ilinca, National Research Council, 75 boul de Mortagne, Boucherville, QC, J4B 6Y4, Canada.

†E-mail: florin.ilinca@cnrc-nrc.gc.ca

‡Research Officer, Industrial Materials Institute

solid boundary. This challenge can be overcome by the use of stabilized finite elements, which provide stable accurate solutions even on highly stretched or distorted meshes. The imposition of the boundary conditions on the immersed boundary is also a point of concern.

In most IB methods, boundary conditions on immersed surfaces are handled either accurately by using dynamic data structures to add/remove grid points as needed [4], or in an approximate way by imposing the boundary conditions to the grid point closest to the surface or through least-squares. In our recent work [5, 6], we proposed an immersed boundary-body conformal enrichment (IB-BCE) method that achieves the level of accuracy of cut cell dynamic node addition techniques with none of their drawbacks (increased CPU time and costly dynamic data structures). The discretization of elements cut by the fluid/solid interface is enriched by the addition of DOFs associated to fluid/solid interface nodes, which are later eliminated at element level.

The goal of the present work is the solution by means of the IB-BCE FEM proposed in [5, 6] of flows in presence of complex-shaped solid surfaces as those encountered in the case of metallic foam matrices. The combination of computer simulations and X-Ray Computed Tomography (CT) for the study of porous media was initiated in the 1990s. The X-Ray CT can provide detailed material structure, which can then be used in computer simulations. Direct calculations on reconstructed images of porous micro-structure from X-ray computed micro-tomography has been shown capable of predicting a range of properties which are important in many sectors. In particular, it has been shown that direct calculations on high-resolution images of Fontainebleau sandstone can produce permeability and conductivity values which are in very good agreement with laboratory measurements [7–10]. Direct computations at the pore scale using tomographic images are now being used in many applications [11].

The X-ray tomography is being used in the area of metallic foams to investigate micro-structure and determine pore size and similar geometric parameters [12, 13]. However, not so many studies involve direct flow simulation at pore scale level of open-cell metallic foams. Vicente *et al.* [14] used 3D tomographic data to determine physical properties of nickel–chromium metallic foams. They proposed morphological analysis methods that allow quantitative measures of the foam structure. Furthermore, they conducted conductive heat transfer inside the metallic structure of the foam. Petrasch *et al.* [15] analyzed the permeability of a 10-ppi (pores per inch) reticulated porous ceramic from the Stokes flow regime to the unsteady flow regime. They reconstructed the 3D foam micro-structure geometry and conducted 3D incompressible fluid flow simulations for Reynolds number from 0.2 up to 200. Flow simulations were conducted using a finite volume method (ANSYS-CFX) with unstructured, body-fitted, tetrahedral mesh discretization. They compared the predicted permeability coefficients to the values determined by selected porous media flow models. The proposed method achieved good agreement with most of the other flow models. Magnico [16] studied hydrodynamic properties of metallic foams at the pore scale on a microtomographed sample from creeping flow to unsteady inertial flow. Both finite volume and Lattice Boltzmann methods were used and validated against experimental data. They solved the incompressible Navier–Stokes equations using a second order, under-relaxed, SIMPLE finite volume formulation. No-slip boundary conditions were applied at the fluid–solid interface, and a prescribed pressure drop was imposed on a periodic computational domain. The method was validated against known solutions. In the work of Calvo *et al.* [17], X-ray micro-tomography and X-ray radiography were used to investigate single and two-phase flows in Ni–Cr metallic foams. They measured the pressure drop through Racemat RCM-NCX-1116 metallic foam for air velocities from 0.02 to 1.34 m/s. Pore scale predictions were conducted using the Lattice Boltzmann method [18] and compared with experiments as well as with other models proposed in the literature. Good agreement was achieved for the flow regimes tested.

In this work the flow inside a metallic foam matrix is solved using a multiscale approach combining the solution at the pore level by the immersed boundary method and the macro-scale solution with simulated permeability. The paper is organized as follows. First, the model problem and the associated finite element formulation is presented. The IB-BCE formulation and the procedure to impose the boundary conditions on the fluid/solid interface are discussed. Section 3 illustrates the performance of the present IB-BCE method for a selection of 3D test problems. The method is then applied to the solution of the flow through a metallic foam matrix. The computed pressure drop, as

a function of the flow rate for two experimental setups, is compared with the experimental data for various foam thicknesses.

2. FLOW MODELING

2.1. Model equations and boundary conditions

The equations of motion are the steady-state incompressible Navier–Stokes equations:

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nabla \cdot \left[\mu \left(\nabla \mathbf{u} + (\nabla \mathbf{u})^T \right) \right] + \mathbf{f} \quad (1)$$

$$\nabla \cdot \mathbf{u} = 0, \quad (2)$$

where ρ is the density, $\mathbf{u}$ the velocity vector, p the pressure, μ the dynamic viscosity, and $\mathbf{f}$ a volumetric force vector. The interface Γ_i between the fluid and solid regions is defined using a level-set function ψ , which is defined as a signed distance function from the immersed interface:

$$\psi(\mathbf{x}) = \begin{cases} d(\mathbf{x}, \mathbf{x}_i), & \mathbf{x} \text{ in the fluid region,} \\ 0, & \mathbf{x} \text{ on the fluid/solid interface,} \\ -d(\mathbf{x}, \mathbf{x}_i), & \mathbf{x} \text{ in the solid region,} \end{cases} \quad (3)$$

where $d(\mathbf{x}, \mathbf{x}_i)$ is the distance between the point $P(\mathbf{x})$ and the fluid/solid interface. Hence, points in the fluid region will take on positive values of ψ , whereas ψ for points in the solid region will be negative. The definition of the level-set function may be more complicated for complex 3D geometries. In such a case, we may consider that the immersed boundary surface is provided in the form of a CAD file from which we generate a surface mesh with a mesh size sufficiently small as to have a correct representation of the surface. Then, the level-set function is simply computed from the shortest distance between the nodes of the 3D mesh and those of the surface mesh.

The boundary conditions associated with the momentum-continuity equations are

$$\mathbf{u} = U_D(\mathbf{x}), \quad \text{for } \mathbf{x} \in \Gamma_D, \quad (4)$$

$$\mu (\nabla \mathbf{u} + \nabla \mathbf{u}^T) \cdot \hat{\mathbf{n}} - p \hat{\mathbf{n}} = \mathbf{t}(\mathbf{x}), \quad \text{for } \mathbf{x} \in \Gamma_t, \quad (5)$$

where Γ_D is the portion of the fluid boundary $\partial\Omega_f$ where Dirichlet conditions are imposed, and $\mathbf{t}$ is the traction imposed on the remaining fluid boundary $\Gamma_t = \partial\Omega_f \setminus \Gamma_D$. Dirichlet boundary conditions are imposed at the interface between fluid and solid regions, i.e. $\Gamma_i \subset \Gamma_D$. Because Γ_i is not represented by the finite element discretization, a special procedure is used to enforce velocity boundary conditions on this surface. This approach is described in the next section.

2.1.1. The immersed boundary-body conformal enrichment method. The flow equations are solved using an FEM coupled with an immersed boundary-body conformal enrichment approach [5]. In the present work, we discretize the entire domain $\Omega = \Omega_f \cup \Omega_s$, where Ω_f and Ω_s denote the fluid and solid regions, respectively. A special treatment is applied when solving inside the solid and in the vicinity of the fluid/solid interface. The mesh is intersected by the interface Γ_i at points located along element edges, and we consider those points as additional DOFs in the finite element formulation. In Figure 1, the mesh is shown by the black continuous lines, and the interface is indicated by the discontinuous line. The proposed technique was implemented for 3D meshes, but for the ease of the presentation, only a 2D case is illustrated in Figure 1. The fluid/solid interface is defined by a level-set function, and therefore, the additional nodes are easily determined as being those for which $\psi = 0$. Moreover, because the level-set function is interpolated using linear shape functions, the intersection between Γ_i and a tetrahedral element is a plane, either a triangle or a quadrilateral (some exceptions are cases when the interface intersects with one or more grid nodes [5]). Elements cut by Γ_i are therefore decomposed into sub-elements which are either entirely in the fluid or entirely in the solid regions. In Figure 1 (Ef) denotes fluid elements, (Es) represents solid elements, whereas interface elements (Ei) are decomposed into fluid sub-elements (Eif) and solid sub-elements (Eis). The original mesh nodes are either inside the fluid (Nf), identified by the

Figure 1. Decomposition of elements cut by the immersed boundary.

green-filled circles, or inside the solid region. Solid nodes are separated into those connected with nodes in the fluid region (N_{sf})—yellow filled circles, and nodes which are not connected with nodes in the fluid region (N_s)—gray filled circles. The additional interface nodes (N_i) are identified by red-filled squares. Once interface elements are decomposed, flow equations are solved in the entire fluid region (i.e., for all (E_f) and (E_{if}) elements), which in Figure 1 is indicated by the gray-filled region of the computational domain. The addition of DOFs associated with the nodes on the interface would normally result in a modification of the global matrix resulting from the finite element equations. In such a case, the implementation is more challenging, and the increase in computational cost is inherent as dynamic data structures, and renumbering are needed. The present approach does not need the explicit addition of the degrees of freedom associated to those nodes. The procedure is described in details in references [5, 6].

2.1.2. Finite element solution. The incompressible Navier–Stokes equations are solved by a Streamline-upwind/Petrov–Galerkin/Pressure Stabilized Petrov–Galerkin (SUPG/PSPG) formulation [19–21]. Stabilization methods such as SUPG and PSPG are built by adding stabilization terms to the Galerkin formulation. The Galerkin part of the variational equations is obtained by multiplying Equations (1)–(2) by appropriate test functions and integrating over the domain of interest. Weak terms are then obtained by using the divergence theorem applied to the momentum diffusion and pressure gradient terms. The SUPG/PSPG method contains additional stabilization terms, which are integrated only on the element interiors. These terms provide smooth solutions to convection dominated flows and deal with the velocity–pressure coupling so that equal-order interpolation results in a stable numerical scheme [19–21]. This makes it possible to use elements that do not satisfy the Babuška–Brezzi condition, as is the case of the linear $P1 - P1$ element [19, 22].

The nonlinear equations for the velocity and pressure are solved with a few Picard steps followed by Newton–Raphson iterations. The resulting linear systems are generated directly in a compressed sparse row format [23] and solved using the bi-conjugate gradient stabilized (Bi-CGSTAB) iterative method [24] with an ILU preconditioner. An important reason for using the SUPG formulation is that it also stabilizes the linear systems, making them tractable by iterative solvers [25]. For more details on the finite element solution algorithm, the reader could consult reference [5].

3. VERIFICATION OF IMMERSED BOUNDARY-BODY CONFORMAL ENRICHMENT METHOD

3.1. Flow around an array of spheres

The IB-BCE method is first verified on the flow around spheres placed on a simple cubic lattice. The flow enters the computational domain at constant velocity U_0 , and the distance between the center of adjacent spheres is L . Simulations were performed for various values of the sphere diameter D ranging between $0.2L$ and $1.3L$, resulting in a solid fraction of the bed of spheres varying between 4.19×10^{-3} and 0.896 . Following the work of Martys *et al.* [26], computations were carried out

in low Reynolds number regime ($Re = \rho U_0 L / \mu \ll 1$) for which the inertia in the momentum equations can be neglected. Hence, flow equations can be written in the following dimensionless form:

$$0 = -\nabla^* p^* + \nabla^* \cdot [\nabla^* \mathbf{u}^* + (\nabla^* \mathbf{u}^*)^T] \quad (6)$$

$$\nabla^* \cdot \mathbf{u}^* = 0, \quad (7)$$

where $\nabla^* = L \cdot \nabla$, $\mathbf{u}^* = \mathbf{u} / U_0$, $p^* = p / p_0$, and $p_0 = \mu U_0 / L$.

Given the symmetry of the arrangement of spheres, only one array is considered having the size of the transverse section equal to the distance between the centroids of two adjacent spheres. Computations were carried out for one array of five spheres in the flow direction. The computational domain spans in the flow direction from $x = -4L$ to $x = 9L$ with the five spheres located in the space between $x = 0$ and $x = 5L$ (the centroids are at $x = 0.5L$, $x = 1.5L$, $x = 2.5L$, $x = 3.5L$, $x = 4.5L$, respectively). The domain size is equal to L in y and z directions. Because the boundary planes normal to the flow direction at $y = \pm L/2$ and $z = \pm L/2$ are symmetry planes, we impose slip, non-penetrating boundary conditions for the velocity at these locations. The surface of the spheres on which no-slip boundary conditions are imposed is represented by the level-set function:

$$\psi(x, y, z) = \min(\psi_0, ((x - x_i)^2 + (y - y_i)^2 + (z - z_i)^2)^{1/2} - D/2), \text{ for } i = 1, N_s \quad (8)$$

where N_s is the number of spheres in the array, ψ_0 is an arbitrary positive value larger than the size of the computational domain (say $\psi_0 = 100L$) and (x_i, y_i, z_i) are the centroid coordinates of the sphere i . The level-set $\psi = 0$ will then indicate the location of the solid boundary. The region having $\psi > 0$ is the flow region inside which the flow equations are solved, and the region having $\psi < 0$ represents the volume of the spheres. The solid fraction of the bed of spheres is given by the following:

$$f_s = V_s / V_{tot} \quad (9)$$

where V_s is the volume occupied by one sphere

$$V_s = \frac{\pi D^3}{6}, \quad \text{for } D \leq L \quad (10)$$

$$V_s = \frac{\pi D^3}{6} - 6 \frac{\pi D^3}{12} + 6 \frac{\pi L}{24} (3D^2 - L^2), \quad \text{for } D > L \quad (11)$$

and $V_{tot} = L^3$ is the volume of the cubic box of size L within which the sphere is located. The expression in Equation (11) takes into account the spheres overlapping when $D > L$.

The behavior of the IB-BCE method for this test problem was also investigated in our previous work [5]. Here, we are interested in verifying the accuracy of the method with respect to mesh refinement and the sensitivity to small changes in the position of the fluid/solid interface. Computations were carried out for three meshes whose characteristics are given in Table I. A cut through Mesh 1 corresponding to the mid-plane in the flow direction is shown in Figure 2 for a diameter $D = 0.7L$. The mesh has 20 elements in directions transverse to the flow. As can be seen, the solid boundary of the spheres cuts through the elements, as the mesh is not constructed to fit the geometry of the immersed objects.

Table I. Mesh characteristics for flow around an array of spheres.

Mesh	Element size	# of nodes	# of elements
1	$L/20$	62,181	280,000
2	$L/30$	202,181	940,000
3	$L/40$	472,361	2,240,000

Figure 2. Mesh 1 for $D = 0.7L$.

The permeability k of the array of spheres is determined from the Darcy's law and is given as follows:

$$k = \frac{\mu U_0}{|\nabla p|} \quad (12)$$

where the pressure gradient is determined as $\nabla p = (p_e - p_i)/(N_s L)$ with p_e and p_i representing the computed pressure at the domain exit and inlet, respectively, and N_s being the number of spheres in the array (in the present case $N_s = 5$).

The solution for the permeability of the array of spheres using Mesh 3 is shown in Figure 3 as a function of the solid fraction. The results are compared with those of Chapman and Higdon [27]. Similar results were reported by Martys *et al.* [26] using a lattice Boltzmann method. As can be seen, the permeability decreases as the solid fraction increases, reaching a value close to 0 when the solid fraction approaches the maximum value of 1. The present numerical results are in excellent agreement with those reported by Chapman and Higdon [27].

One characteristic of the present IB-BCE method is the fact that boundary conditions on the fluid/solid interface are imposed with the same accuracy as if the mesh was body-conformal. Thus, the method is very sensitive to the position of the solid boundary inside mesh elements. Note that because the FEM uses linear interpolation functions, the boundary is represented by planar surfaces inside mesh elements. However, it would have been the same using a body-conformal mesh. Thus, the accuracy with which the boundary is represented depends on the local mesh size. In order to assess the sensitivity of the IB-BCE method to changes in the location of the solid boundary within a mesh element, computations were carried out by varying the spheres diameter from $0.7L$ to $0.8L$ with a constant increment of $0.01L$. The results are plotted in Figure 4(a). Furthermore, very small changes in the location of the solid boundary were simulated by considering the diameter in the range between $0.7L$ and $0.71L$ with constant increments of $0.001L$. Recall that the mesh size is $0.05L$ for Mesh 1 and $0.025L$ for Mesh 3, which means that each change in the radius of the spheres determines a displacement of the interface with a distance 50 to 100 times smaller than the element size. The results from this set of computations are shown in Figure 4(b). As can be seen, the present numerical method is able to represent accurately even extremely small changes in the location of the immersed boundary.

Figure 3. Permeability of an array of spheres.

Figure 4. Permeability for small changes in sphere diameter. (a) diameter increments of $0.01L$ ($D/L = 0.7 \div 0.8$); (b) diameter increments of $0.01L$ ($D/L = 0.7 \div 0.71$).

The high sensitivity of the present IB-BCE method to the position of the interface gives us the possibility to compute numerically the derivative of the solution, in this case the permeability, with respect to sphere diameter. We propose to use the following central finite differences to compute the permeability derivative with respect to D :

$$\frac{\partial k}{\partial D}(D_0) = \frac{k(D_0 + \delta D) - k(D_0 - \delta D)}{2\delta D} \quad (13)$$

where δD is a small perturbation around D_0 . In the following, we considered $\delta D = 0.01D_0$. The results obtained for the three meshes are compared in Figure 5(a). As can be seen the derivative estimated on Mesh 1 is slightly less accurate for $D = 0.3L$, whereas those obtained on meshes 2 and 3 are practically the same. One use of the derivative is the fast computation of nearby solutions. In the present case, one can compute the permeability for values of the diameter around a nominal value D_0 by using the following:

$$k(D_0 + \delta D) = k(D_0) + \frac{\partial k}{\partial D} \delta D \quad (14)$$

The nearby solutions for the permeability are illustrated in Figure 5(b). The solution at the nominal values of the diameter are indicated by the open circles, whereas the nearby solutions are illustrated by the thicker line. We can see that the use of the solution derivative is reliable, the variation of the

Figure 5. Derivative and nearby solutions for the permeability. (a) computed derivative $\frac{\partial k}{\partial D}$; (b) nearby solution for permeability.

permeability indicated by the nearby solutions being in excellent agreement with the general behavior of the permeability curve. The ability to compute the derivative of the solution with respect to the position of the interface opens the way to use gradient based optimization algorithms in the context of optimal design.

3.2. Flow around alternated arrays of cylinders

The second validation problem represents the flow around alternated arrays of cylinders. Cylinders of diameter D and infinite length are placed at a distance L from each other. Their orientation is normal to the flow direction but alternates between vertical (along y axis) and horizontal (along z axis) from one array to the next as shown in Figure 6. In addition to the direction change between successive layers, the position of cylinders changes between odd and even vertical/horizontal layers, such as the unit cell of the configuration is as shown in Figure 6(b). The unit cell has the dimensions $4L \times L \times L$ and the configuration shown in Figure 6(a) contains three cells in x direction, three cells in y and four cells in z . The cylinders' position inside the arrangement is detailed in Table II, the four rows being replicated in x direction as many times as the number of cells in the respective direction. This configuration is such as to replicate the features of a metallic foam matrix with the distance L between two cylinders being the equivalent of the pore size of the foam. The arrangement of cylinders is such as to avoid having an easy flow path around the solid structure and also not to be too complex to generate. This test problem can be used to investigate the evolution of the permeability as a function of the flow regime (Re number), as a function of the solid fraction and as a function of the number of cells considered in the flow direction. The solid fraction of the alternating cylinders arrangement can easily be changed by modifying the diameter of the cylinders. Remark that one important advantage of the IB-BCE method is that solutions for various values of the diameter can be obtained on the same mesh by only changing the level-set function used to locate the fluid/solid interface.

In this study, we were interested only in the flow regimes for which the solution is steady state, that means, for $Re < Re_{cr}$, with Re_{cr} being the Reynolds number for which the flow becomes unsteady. As a result, the flow is periodic for cells in the y and z directions, and there is no need to consider more than one cell in those two directions. As for the previous validation test, the boundary planes normal to the flow direction are symmetry planes on which we impose slip, non-penetrating boundary conditions for the velocity. To study the flow dependence on the number of cells in the x

Figure 6. Alternating arrays of cylinders (a) $3 \times 3 \times 4$ cells and (b) one single cell.

Table II. Position of each cylinder in the alternating arrays arrangement.

Row	x/L	y/L	z/L
1	0.5	∞	..., 0.5, 1.5, 2.5, ..
2	1.5	..., 0.5, 1.5, 2.5, ..	∞
3	2.5	∞	..., 0, 1, 2, ..
4	3.5	..., 0, 1, 2, ..	∞

direction, we considered the configurations having one, two, three, and four cells in the flow direction. The configuration with one cell (C1) consists of four rows of cylinders, the configuration with two cells (C2) has eight rows of cylinders and so on.

A first series of computations were carried out to determine the correlation between the pressure gradient and the Reynolds number. We consider $Re = \rho UL / \mu$, where ρ is the fluid density, U is the fluid velocity in the uniform flow in front of the cylinders and μ is the fluid dynamic viscosity. Simulations were performed considering one cell in the flow direction and for five different meshes with gradually smaller mesh element size as shown in Table III. Both computations using the IB-BCE method and using body-conformal meshes are carried out.

Configuration C1 was solved for a cylinder diameter $D = 0.4L$ corresponding to a solid fraction $f_s = 0.1256$. Computations were carried out for Re ranging from 0.05 to 200. The results for the pressure gradient are shown in Figure 7 in dimensionless form with $p^* = p/p_0$, $p_0 = \mu U/L$ and $x^* = x/L$. The error with respect to the solution on the finer mesh is shown in Figure 8. As can be seen, if one considers the solution on Mesh 5 as a reference, the error decreases when decreasing the mesh size for both BC and IB-BCE solutions. Note that the improvement is significant from one mesh to the next, except for the body-conformal mesh 3. This behavior is caused by the fact that whereas all meshes are composed of tetrahedrals, IB-BCE meshes are quasi-structured, whereas BC ones are entirely unstructured. Thus we may expect that changes in IB-BCE solutions from one mesh to the next to reflect better the decrease in the mesh element size. Observe also that the solution error increases when increasing Re . On the coarser mesh the error varies from about 2% at low Re to close to 10% for $Re = 200$. The error dependence on Re is smaller for the Mesh 4 for which errors range between 0.25% and 0.75%. The IB-BCE and BC solutions obtained on the finer meshes are compared in Figure 9(a) indicating that the two methods produce similar results. Hence, only the IB-BCE method will be used in the remaining of this investigation.

The pressure gradient in the flow direction as a function of the flow rate is very important for porous media as it characterizes the permeability of the respective media. The most widely

Table III. Mesh characteristics for flow around alternating arrays of cylinders.

Mesh	Element size	# of nodes, IB-BCE / BC	# of elements, IB-BCE / BC
1	$L/20$	81,585 / 76,896	368,000 / 371,825
2	$L/25$	138,580 / 127,055	637,500 / 634,820
3	$L/30$	216,225 / 196,489	1,008,000 / 999,918
4	$L/40$	425,293 / 376,378	2,016,000 / 1,983,168
5	$L/50$	762,093 / 700,276	3,650,000 / 3,785,686

Figure 7. Pressure gradient as a function of Re ; (a) body-conformal mesh solution and (b) immersed boundary-body conformal enrichment method.

Figure 8. Error in $(\partial p^*/\partial x^*)$ with the solution on mesh 5 as reference; (a) body-conformal mesh solution and (b) immersed boundary-body conformal enrichment method.

Figure 9. Comparison of body-conformal and immersed boundary-body conformal enrichment solutions; (a) pressure gradient and (b) derivative of $\partial p^*/\partial x^*$ with respect to Re .

used correlation is the Forchheimer's equation representing an empirical relationship between the pressure gradient in the flow direction and the mean flow speed U :

$$\frac{\partial p}{\partial x} = \frac{\mu}{k_1} U + \frac{\rho}{k_2} U^2 \quad (15)$$

where k_1 and k_2 are permeability coefficients. In dimensionless form the Forchheimer's equation is written as:

$$\frac{\partial p^*}{\partial x^*} = \frac{1}{k_1^*} + Re \frac{1}{k_2^*} \quad (16)$$

with $k_1^* = k_1/L^2$ and $k_2^* = k_2/L$. Hence, in the Forchheimer model, the dimensionless pressure gradient is a linear function of Re . The coefficient k_1^* is then determined from the pressure gradient at $Re = 0$. The resulting values as from the solution on the finer meshes are $k_1^* = 0.03129$ for the IB-BCE solution and $k_1^* = 0.03116$ for the BC solution, the difference between them being of 0.4%. A way to estimate the second permeability coefficient k_2^* , also called inertial coefficient, is to estimate the slope of $\partial p^*/\partial x^*$ as a function of Re . In the ideal case, which would obey strictly to the Forchheimer model, the slope of the curve would be a constant. An estimation of the derivative of the pressure gradient with respect to Re can be obtained numerically by using central finite differences. The results obtained when using $\delta Re = 0.02 Re$ are shown in Figure 9(b). Here again, the

IB-BCE and BC solutions are in agreement with each other. We observe that the inertial coefficient is close to zero for low Reynolds number flows. Therefore, in this regime, the Darcy's law is a better approximation of the flow behavior than the Forchheimer model. The inertial coefficient increases gradually with the increase in Re reaching the maximum value near $Re = 50$ and then starts to decrease when increasing Re . We may consider that for $Re > 20$, the inertial coefficient can be approximated by a constant, which in this case would be $k_2^* \approx 2.0$.

The dependence of pressure gradient on the number of cells considered is investigated by solving configurations having one, two, three and four cells (C1, C2, C3, and C4 respectively). Simulations were performed using a mesh with $L/30$ element size, which was identified as Mesh 3 for case C1. The solutions C1, C2 and C3 are compared in Figure 10(a) indicating very small differences when increasing the number of cells in the flow direction. The error in the pressure gradient with respect to the solution for four cells (C4) is shown in Figure 10(b). As expected, the error decreases when increasing the number of cells. The maximum error for C1 is about 2% and occurs for $Re = 50$ and for $Re = 200$. The difference with respect to C4 decreases to less than 0.5% for the C3 configuration. This simulations indicate that the error introduced by modeling only one cell instead of a larger number of cells is small (less than 2%) for most values of Re .

The influence of the cylinders' diameter and implicitly of the solid fraction inside the cells was investigated for the C1 configuration and mesh 5. The Reynolds number was taken equal to 5, 50 and 100 respectively, and the diameter was varied from $0.1L$ to $0.7L$ with $0.05L$ increments. The corresponding solid fraction takes values from 7.85×10^{-3} to 0.385. The pressure gradient normalized to the value taken for $D = 0.4L$ is plotted in Figure 11. As expected, the pressure gradient

Figure 10. Influence of the number of cells on the pressure gradient; (a) solutions for various number of cells and (b) error with respect to C4.

Figure 11. Influence of the solid fraction on the pressure gradient.

increases when increasing the solid fraction, with the behavior being more pronounced at higher solid fraction values. Note also that the normalized values depend little on Re indicating that the dependence on the solid fraction is the same in all flow regimes.

4. APPLICATION: FLOW THROUGH A METALLIC FOAM MATRIX

In this application, we are interested in solving the flow of water through an open cell NC-1723 nickel chromium foam having a pore size of 0.9 mm. Experimental measurements for the flow through two different setups and for various thicknesses of the foam ranging from 4.3 mm to 38.6 mm were reported by Lefebvre *et al.* [28].

Numerical investigation of this flow problem is carried out in two steps. First, the 3D Navier–Stokes equations describing the fluid flow at pore level are solved on small samples of the foam. The goal of this model is to predict the average permeability of the foam, which then is used in a second model representing the actual experimental apparatus. In this second step, instead of modeling the geometrical details of the foam, the foam presence is taken into account by incorporating in the flow equations a source term providing the same pressure drop as the one computed at the micro scale. In this way, more complex flow configurations in which the foam is only part of the computational domain can be considered. This is carried out without solving an overly large problem, as it would have been the case by tacking into account all the geometrical details of the foam.

4.1. Micro-scale simulations

For the micro-scale simulations, the metallic foam surface of small samples having $3.03 \text{ mm} \times 3.03 \text{ mm} \times 4.04 \text{ mm}$ is determined using micro-CT scan reconstructions. The scan has a resolution of 0.0101 mm, thus providing data on a $300 \times 300 \times 400$ grid. This information is then used to initialize the level-set function representing the surface of the metallic foam in the IB-BCE approach. Fluid properties are those of water at ambient temperature: density $\rho = 996.1 \text{ kg/m}^3$ and viscosity $\mu = 8.3 \cdot 10^{-4} \text{ Pa} \cdot \text{s}$. The flow is considered in the positive direction of the z -axis and the computational domain extends 2 mm before the foam and 20 mm in the wake after the foam. The domain size in x and y direction is the same as the sample size, and slip no-penetrating velocity boundary conditions are imposed on all external surfaces except for the entry and the exit. This boundary conditions suppose that the foam is formed by mirror reproduction of the selected sample over the volume occupied by the foam. Even if this assumption considers that the entire foam has the same structure as the selected sample, we take into account the changes in the foam structure by considering separately micro-scale simulations of samples located at various locations inside the foam. A uniform velocity is imposed on the entry surface and free traction conditions at exit. The mesh in the region of the foam sample is $75 \times 75 \times 100$, thus having 2,812,500 tetrahedral elements in the region of the foam. The entire mesh including that covering the regions before and after the foam sample has 722,000 nodes and 3,487,500 elements. Remark that the scanning grid of the micro-CT is four times finner than the computational grid in all directions. The initialization of the level-set function used all scan data such as interpolations inside interface elements to result in a position of the interface as close as possible to the measured one.

Computations were performed for five different foam samples, and the inlet velocity was taken between $U = 8.3 \times 10^{-5} \text{ m/s}$ and $U = 0.38 \text{ m/s}$. The corresponding Reynolds number based on the pore size $L = 0.9 \text{ mm}$ varies between $Re = 0.09$ and $Re = 410$. The critical Reynolds number Re_{cr} marking the transition from a steady state flow towards unsteady solutions is expected to be around $Re_{cr} \approx 200$. The boundary conditions used in the micro scale simulations are correct if we assume that the foam is composed of cells having the size of a sample, which are replicated in all directions to cover the volume of the entire foam. Even in such a case, the slip non-penetrating boundary conditions assume also that the flow is steady state, as unsteady flows will present flow exchanges between the cells in direction normal to the flow. Numerical simulations indicate that the transient effect for Re in the range 200 to 400 is small and has little impact on the pressure drop when compared with steady state solutions in the same flow regime. Thus, we extended the micro-scale simulations to flows at Re up to 410.

The pressure distribution on the surface of the foam and selected flow streamlines for samples S1 and S3 are shown in Figure 12. The pressure is higher at the flow inlet and is set to zero on the outlet. Higher pressures are computed on the parts of the foam surface, where the flow impacts and lower pressure is determined in the wake of each single foam structure. Streamlines indicate that the foam determines an important mixing of the flow even if samples of relatively small sizes are considered.

The dependence of the pressure gradient across the foam with respect to Re is shown in Figure 13 for the five samples considered. Here again, the results are given in the dimensionless form used in Equation (16). Hence, the pressure gradient at $Re = 0$ is the inverse of the permeability coefficient k_1^* , and the slope of the curve is the inverse of the inertial coefficient k_2^* . The resulting permeability coefficients for all five samples are given in Table IV. Results indicate that all samples have similar permeability coefficient k_1^* . Three samples (S1, S2 and S4) have almost the same k_2^* , whereas sample S3 presents the smaller pressure drop (larger inertial permeability) and sample S5 the larger pressure drop (smaller permeability). The results for the five samples are not too far from each other indicating that the size of the samples is adequate in providing the permeability of the foam.

Figure 12. Pressure distribution and streamlines for samples (a) S1 and (b) S3.

Figure 13. Variation of the pressure gradient with Re for different foam samples.

Table IV. Permeability coefficients as from the IB-BCE solution at micro scale.

Sample	k_1^*	k_2^*
S1	28.86×10^{-3}	1.178
S2	28.37×10^{-3}	1.183
S3	28.59×10^{-3}	1.250
S4	26.56×10^{-3}	1.211
S5	25.02×10^{-3}	1.067

As shown in the previous applications, the IB-BCE method has the advantage that can lead to a rigorous study of the dependence of the flow solution on the position of the interface. In the present case, one can determine how the permeability coefficients would change if the solid fraction of a foam sample changes. Simulations for the foam configuration of sample S1 were carried out by using a different threshold for the location of the fluid/solid interface in the μ -CT scan data. Thus, the corresponding solid fraction of the foam varies between 10.3% and 12.5%. The resulting permeability coefficients k_1^* and k_2^* are compared with the solutions for the four remaining samples in Figure 14. The solutions for the sample S1 at various solid fractions (open triangles in Figure 14) indicate, as expected, a tendency for lower permeability at higher solid fraction. The same behavior is observed when comparing the various samples (filled squares in Figure 14) especially for the inertial coefficient k_2^* . Samples S4 and S5 present a lower permeability coefficient k_1^* than sample S1 at similar solid fraction.

4.2. Macro-scale simulations

The micro scale simulations provide the pressure drop as a function of the flow rate and hence the permeability coefficients of the foam. This information is then used to perform macro-scale simulations for which, instead of modeling the detailed foam geometry, we include in the momentum equations a source term producing the same pressure gradient as the one predicted by the micro-scale solution and being opposite to the flow:

$$\mathbf{f} = - \left(\frac{\mu}{k_1} \mathbf{U} + \frac{\rho}{k_2} \mathbf{U}^2 \right) \frac{\mathbf{u}}{U} \quad (17)$$

where U is the norm of the velocity vector $\mathbf{u}$. This source term is applied in the entire volume occupied by the foam.

We present here numerical results for the setups shown in Figure 15 for which experimental data are available [28]. The ratio between the surface of the nominal flow section prior to the foam to the surface of the sample is 0.87 for setup A and 0.29 for setup B. The foam thicknesses for which experimental data are available are 4.27 mm, 8.6 mm, 13.7 mm, 21.0 mm, 28.2 mm, and 38.6 mm.

The computational domain for setup A extends 120 mm upward from the foam location and 120 mm downward from the foam. The respective dimensions for the setup B computation are 100 mm and 200 mm. The inlet velocity is uniform and ranges between 0.824×10^{-4} and 0.375 m/s, as was the case for the micro-scale computations. Simulations were carried out by using the permeability coefficients obtained on samples 1, 3 and 5. Recall that samples 2 and 4 have similar permeability as sample 1. The pressure, velocity, and streamlines in a plane through the symmetry axis computed with the permeability coefficients of sample 1 are shown in Figure 16 for the case

Figure 14. Permeability coefficients as a function of the solid fraction; (a) permeability coefficient, k_1^* and (b) inertial coefficient, k_2^* .

Figure 15. Details of the two experimental setups, (a) A and (b) B, used to fix the specimens.

Figure 16. Pressure, velocity and streamlines for $L = 38.6$ mm; (a) pressure for setup A, (b) pressure for setup B, (c) velocity and streamlines for setup A, and (d) velocity and streamlines for setup B.

$L = 38.6$ mm and an inlet velocity of 0.305 m/s. Solutions indicate that, as expected, larger pressure gradients are obtained in the region of the foam. Remark that the change in flow section at the entry and exit of the foam determines a deviation of the streamlines from the straight parallel flow. This effect is more important in the setup B for which the difference between the flow section and the foam section is larger.

The pressure drop determined by the presence of the foam was calculated between two locations at 50 mm on each side of the foam. The numerical results are compared with the experimental data in Figure 17. As can be seen, the agreement between the simulation and experiment improves when increasing the thickness of the foam. The agreement is very good for the thickest foam (Figure 17(e) for setup A and Figure 17(f) for setup B).

The computed pressure drop for various foam thicknesses and flow speed are shown in Figure 18. The numerical solutions confirm several experimental observations. The pressure drop depends on the way the setup is designed, and the pressure drop dependence on the sample thickness decreases as the ratio of the foam section exposed to the flow to the total foam section approaches 1. The pressure gradient sensitivity to the foam thickness is smaller for the setup A than that for the setup B. These lateral diffusion effects, which are more important in the setup B, are verified by the experimental observations.

5. CONCLUSION

This work presents applications of an immersed boundary FEM to flow in the presence of complex-shaped surfaces. Application to the flow around an array of spheres indicated that the proposed method is accurate and very sensitive to the position of the fluid/solid interface. The sensitivity of the solution to the spheres diameter was accurately determined and used to fast computation of nearby solutions.

Figure 17. Comparison of computed and measured pressure gradients; (a) setup A, $L = 8.6$ mm, (b) setup B, $L = 8.6$ mm, (c) setup A, $L = 21.0$ mm, (d) setup B, $L = 21.0$ mm, (e) setup A, $L = 38.6$ mm, and (f) setup B, $L = 38.6$ mm.

The IB-BCE method was then validated against solutions on body-conformal meshes for the flow around alternated arrays of cylinders replicating the features of a metallic foam matrix. Solutions were computed for meshes with various levels of refinement indicating a good agreement between the two approaches. The IB-BCE method was used to analyze the dependence of the permeability on the flow speed, the solid fraction, and the number of cells in the flow direction. The following observations can be drawn from these numerical tests: (a) the error on coarser meshes is larger at

Figure 18. Solutions for the setups (a) A and (b) B for different foam thickness.

higher Reynolds number; (b) when changing the diameter of the cylinders, the pressure gradient varies in the same way for all flow regimes, and (c) the error when solving one cell when compared with the solution on four cells is less than 2%.

Finally, a multi-scale methodology for characterizing numerically the flow permeability in metallic foams is presented. The proposed approach considers both the foam details at the pore level and the actual behavior when placed inside more complex flow configurations. Simulation results agree well with experimental data, thus opening the way for more extensive numerical studies of the flow inside metallic foams. Such approach could be used for the design of new applications involving flow through porous materials.

ACKNOWLEDGEMENTS

The authors wish to thank Eric Baril and Jean-Philippe Marcotte from the National Research Council Canada for their contribution in generating the micro-CT data of the metallic foam.

REFERENCES

1. Mittal R, Iaccarino G. Immersed boundary methods. *Annual Review of Fluid Mechanics* 2005; **37**:239–261.
2. Löhner R, Cebal JR, Camelli FE, Appanaboyina S, Baum JD, Mestreau EL, Soto OA. Adaptive embedded and immersed unstructured grid techniques. *Computer Methods in Applied Mechanics and Engineering* 2008; **197**:2173–2197.
3. Löhner R, Appanaboyina S, Cebal JR. Comparison of body-fitted, embedded and immersed solutions of low Reynolds-number 3-D incompressible flows. *International Journal for Numerical Methods in Fluids* 2008; **57**: 13–30.
4. Noble DR, Newren EP, and Lechman JB. A conformal decomposition finite element method for modeling stationary fluid interface problems. *International Journal for Numerical Methods in Fluids* 2010; **63**:725–742.
5. Ilinca F, Héty J-F. A finite element immersed boundary method for fluid flow around rigid objects. *International Journal for Numerical Methods in Fluids* 2011; **65**:856–875.
6. Ilinca F, Héty J-F. A finite element immersed boundary method for fluid flow around moving objects. *Computers and Fluids* 2010; **39**:1656–1671.
7. Spanne P, Thovet J, Jacquin C, Lindquist W, Jones K, Adler P. Synchrotron computed microtomography of porous media: topology and transports. *Physical Review Letters* 1994; **73**:2001–2004.
8. Auzerais F, Dunsmuir J, Ferréol B, Martys N, Olson J, Ramakrishnan T, Rothman D, Schwartz L. Transport in sandstone: a study based on three dimensional microtomography. *Geophysical Research Letters* 1996; **23**:705–708.
9. Martys N, Hagedorn J, Goujon D, Devaney J. Large scale simulations of single and multi-component flow in porous media. *Proceedings of SPIE Conference on Developments in X-Ray Tomography II* 1999; **3772**:205–213.
10. Arns J, Arns C, Sheppard A, Sok R, Knackstedt M, Pinczewski W. Relative permeability from tomographic images; effect of correlated heterogeneity. *Journal of Petroleum Science and Engineering* 2003; **39**:247–259.
11. Moreno-Atanasio R, Williams RA, Jia X. Combining X-ray microtomography with computer simulation for analysis of granular and porous materials. *Particuology* 2010; **8**:81–99.

12. Olurin O, Arnold M, Kömer C, Singer R. The investigation of morphometric parameters of aluminum foams using micro-computed tomography. *Materials Science and Engineering A* 2002; **328**:334–343.
13. Bonnet J-P, Topin F, Tadriss L. Flow laws in metal foams: compressibility and pore size effect. *Transport in Porous Media* 2008; **73**:233–254.
14. Vicente J, Topin F, Daurelle J-V. Open celled material structural properties measurement: from morphology to transport properties. *Materials Transactions* 2006; **47**:2195–2202.
15. Petrasch J, Meier F, Friess H, Steinfeld A. Tomography based determination of permeability, Dupuit–Forchheimer coefficient and interfacial heat transfer coefficient in reticulate porous ceramics. *International Journal of Heat and Fluid Flow* 2008; **29**:315–326.
16. Magnico P. Analysis of permeability and effective viscosity by CFD on isotropic and anisotropic metallic foam. *Chemical Engineering Science* 2009; **64**:3564–3575.
17. Calvo S, Beugre D, Crine M, Léonard A, Marchot P, Toye D. Phase distribution measurements in metallic foam packing using X-ray radiography and micro-tomography. *Chemical Engineering and Processing: Process Intensification* 2009; **48**:1030–1039.
18. Beugre D, Calvo S, Dethier G, Crine M, Toye D, Marchot P. Lattice Boltzmann 3D flow simulations on a metallic foam. *Journal of Computational and Applied Mathematics* 2010; **234**:2128–2134.
19. Hughes TJR, Franca LP, Balestra M. A new finite element formulation for computational fluid dynamics: V. circumventing the Babuška–Brezzi condition: a stable Petrov–Galerkin formulation of the Stokes problem accommodating equal-order interpolations. *Computer Methods in Applied Mechanics and Engineering* 1986; **59**:85–99.
20. Hughes TJR, Franca LP, Hulbert GM. A new finite element formulation for computational fluid dynamics: VII. the Galerkin–Least-Squares method for advective-diffusive equations. *Computer Methods in Applied Mechanics and Engineering* 1989; **73**:173–189.
21. Franca LP, Frey SL. Stabilized finite element methods: II the incompressible Navier–Stokes equations. *Computer Methods in Applied Mechanics and Engineering* 1992; **99**:209–233.
22. Tezduyar TE, Mittal S, Ray SE, Shih R. Incompressible flow computations with stabilized bilinear and linear equal-order-interpolation velocity-pressure elements. *Computer Methods in Applied Mechanics and Engineering* 1992; **95**:221–242.
23. Saad Y. *Iterative Methods for Sparse Linear Systems*. PWS Publishing Company: Boston, MA, 1996.
24. Van der Vorst H. Bi-CGSTAB: a fast and smoothly converging variant of Bi-CG for the solution of nonsymmetric linear systems. *SIAM Journal of Scientific and Statistical Computing* 1992; **13**:631–644.
25. Ilinca F, Hétu J-F, Audet M, Bramley R. Simulation of 3-D Mold-Filling and Solidification Processes on Distributed Memory Parallel Architectures. *Proceedings of the ASME Winter Annual Meeting on Fluids Engineering Division*, Vol. 244; Dallas, TX, 1997; 477–484.
26. Martys NS, Hagedorn JG, Goujon D, Devaney JE. Large Scale Simulations of Single and Multi-Component Flow in Porous Media. *Proceedings of the SPIE Conference on Developments in X-Ray Tomography II*, Vol. 3772; SPIE, 1999; 205–213.
27. Chapman AM, Higdon JLL. Oscillatory stokes flow in periodic porous media. *Physical Fluids A* 1992; **4**:2099–2116.
28. Innocentini MDM, Lefebvre LP, Meloni RV, Baril E. Influence of sample thickness and measurement set-up on the experimental evaluation of permeability of metallic foams. *Journal of Porous Materials* 2010; **17**:491–499.