



HAL
open science

Pore-scale and average model numerical simulations of flow in a porous-fluid system

Ch H Bruneau, Didier Lasseux, F J Valdés-Parada

► **To cite this version:**

Ch H Bruneau, Didier Lasseux, F J Valdés-Parada. Pore-scale and average model numerical simulations of flow in a porous-fluid system. 14èmes Journées d'Etude sur les Milieux Poreux, Oct 2018, Nantes, France. hal-02398641

HAL Id: hal-02398641

<https://hal.science/hal-02398641>

Submitted on 7 Dec 2019

HAL is a multi-disciplinary open access archive for the deposit and dissemination of scientific research documents, whether they are published or not. The documents may come from teaching and research institutions in France or abroad, or from public or private research centers.

L'archive ouverte pluridisciplinaire **HAL**, est destinée au dépôt et à la diffusion de documents scientifiques de niveau recherche, publiés ou non, émanant des établissements d'enseignement et de recherche français ou étrangers, des laboratoires publics ou privés.

Pore-scale and average model numerical simulations of flow in a porous-fluid system

Ch.H. Bruneau^a, D. Lasseux^b, F. J. Valdés-Parada^c

^aUniv. Bordeaux IMB, INRIA Bordeaux Sud-Ouest MEMPHIS, CNRS UMR5251, 33405 Talence, Cedex France

^bCNRS UMR5295, I2M Bordeaux, Esplanade des Arts et Métiers, F-33405 Talence, Cedex, France

^cDivisión de Ciencias Básicas e Ingeniería, Universidad Autónoma Metropolitana-Iztapalapa, CDMX 09340, Mexico

Keywords: Fluid-porous media system, Turbulent flow, Numerical simulations

1. Introduction

Inertial (and turbulent) flow over a porous medium, which is of interest in many applications as, for instance, for flow control past a bluff body, is analyzed in this work using numerical modelling. The configuration under consideration is a 2D rectangular domain including a large number of parallel cylindrical particles arranged on an ordered square pattern with a fluid layer of infinite extent at the top, the flow being orthogonal to the cylinders axes. Numerical simulations are carried out at two-different scales. First, pore-scale resolved simulations of the Navier-Stokes equations (DNS) are performed using an accurate finite differences scheme. Second, an average model, based on a one-domain approach, is used to compute the solution within the fluid region and the porous region considered as an equivalent homogeneous medium. This average model corresponds to a Navier-Stokes equation complemented with a Darcy-like term featuring a penalization method for which the apparent permeability in the porous region is modelled on the basis of several different models (namely a first order (K), a second order (K2) and a model making use of the full apparent permeability tensor (H)). Solutions are sought using a multigrid procedure involving several grid levels. The code is highly parallelized with MPI directives. A thorough comparison of the different approaches is proposed through an analysis of the flow fields in the whole domain (inside the porous medium and in the pure fluid zone), highlighting the impact of the penalization approach close to and far from the porous zone.

2. Modelling and results

The whole system is composed of a porous rectangle $R = (100, 500) \times (0, 30)$ immersed in a fluid domain. The total size of the domain including the porous medium is $\Omega = (0, 600) \times (0, 600)$. For the DNS the porous rectangle is made of 12,000 square unit cells (size L_c) with a cylinder of square cross section in the center arranged on a square pattern (30 rows and 400 columns) having a porosity $\epsilon = 0.913$. The flow solution is sought by solving the discretized form of the following equations

$$\partial_t U^* + (U^* \cdot \nabla) U^* - \frac{1}{Re} \Delta U^* + \nabla p^* = 0 \quad (1)$$

$$\partial_t U^* + (U^* \cdot \nabla) U^* - \frac{1}{Re} \Delta U^* + \frac{\epsilon}{Re} H^*(U^*)^{-1} U^* + \nabla p^* = 0 \quad (2)$$

$$\operatorname{div} U^* = 0 \quad (3)$$

where $Re = \frac{\rho U_c L_c}{\mu}$ is the Reynolds number (U_c is the velocity at the infinite upstream). The term $\frac{\epsilon}{Re} H^*(U^*)^{-1} U^*$ is taken as $-\frac{U^*}{K}$ with $K = \frac{\rho k U_c}{\mu \epsilon L_c} = \frac{Re}{\epsilon} \frac{k}{L_c^2}$ (k is the intrinsic permeability of the porous medium) (model K); $-\frac{U^*}{K} + \frac{\epsilon^2 f L_c}{\sqrt{k}} \|U^*\| U^* = \frac{U^*}{K} + \frac{1.75}{\sqrt{150}} \frac{\sqrt{\epsilon} L_c}{\sqrt{k}} \|U^*\| U^*$ taking the value of f given in [1] (model K2); $-\frac{\epsilon}{Re} H^*(U^*)^{-1} U^*$ with the full tensor $H^*(U^*) = H/L_c^2$ (model H). In the latter case, H was computed as a function of Re and the local flow angle [2]. For Ref (*i.e.* DNS), dimensionless equations (1) and (3) are solved in the fluid domain $\Omega_{fT} = \Omega_f \times (0, T)$, Ω_f being the fluid domain included in Ω

and T the simulation time, whereas, for K, K2 and H, equations (1) and (3) are solved in $F_T = F \times (0, T)$, F denoting the free fluid region while equations (2) and (3) are solved in $R_T = R \times (0, T)$.

Solutions are carried using a multigrid method which means that a sequence of uniform grids on Ω are employed, from the coarsest ($G1 = 120 \times 120$ grid blocks) to the finest ($G7 = 7680 \times 7680$, *i.e.* 58982400 grid blocks). Mesh convergence was checked by estimating the error between $G6$ and $G7$. For accuracy, an additional level ($\frac{1}{2}G8$, 132710400 grid blocks) starting from a 90×90 grid blocks level is used for Ref. The simulation time is $T = 3000$.

For $Re = 10$, all the four simulations yield velocity profiles that are superimposed. While increasing Re , (see an example for $Re = 100$ in figure 1), the model H is shown to better perform, although, surprisingly, the model K2 is quite satisfactory in the region close to the exit. Further investigation is under work considering larger values of Re , for which stronger turbulent effects are encountered.

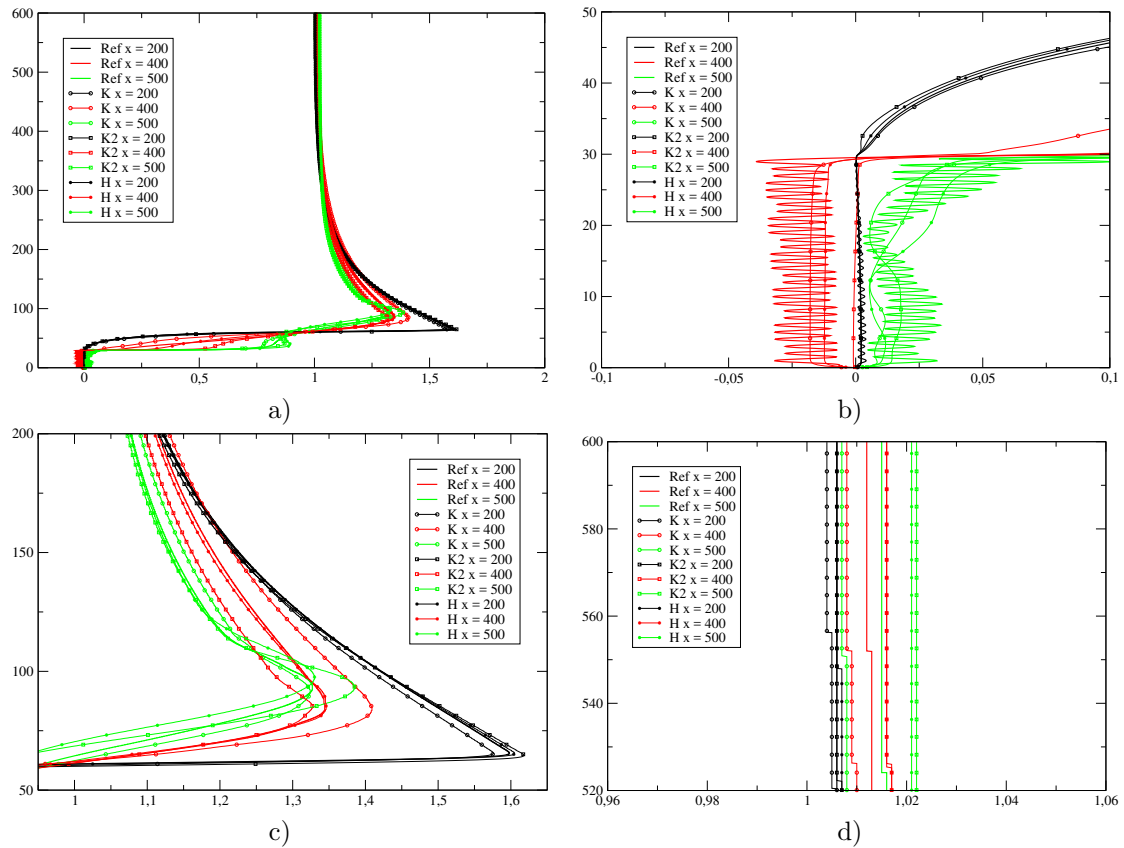


Figure 1: Profiles of the x -component of the velocity at three different vertical section of the domain $x = 200$, $x = 400$ and $x = 500$ for $Re = 100$. a) In the whole sections. b) Inside and just above the porous region. c) In the middle part of the domain. d) At the top of the domain. The three penalization models (K, K2 and H) are compared to the reference flow (Ref) computed on $\frac{1}{2}G8$.

3. Conclusion

Numerical simulations performed on a fluid-porous medium system at large values of the Reynolds number indicate that a penalization method (*i.e.* a one-domain approach), in which the Darcy-like term in the porous domain is pre-computed on the unit cell representative of the structure, provides a quite accurate prediction of the flow, outperforming other types of estimates for this term.

References

- [1] K. Vafai, Convective flow and heat transfer in variable-porosity media. *J. Fluid Mech.*, 147:233-259, 1984.
- [2] D. Lasseux, A. A. Abbasian Arani, A. Ahmadi, On the stationary macroscopic inertial effects for one phase flow in ordered and disordered porous media. *Phys. Fluids*, 23(7), 073103(2011).