A Homogenized Flux-Body Force Approach for Modeling Porous Wall Boundary Conditions in Compressible Viscous Flows

Daniel Z. Huang\textsuperscript{a}, Man Long Wong\textsuperscript{b}, Sanjiva K. Lele\textsuperscript{a,b,c}, Charbel Farhat\textsuperscript{a,b,c}

\textsuperscript{a}Institute for Computational and Mathematical Engineering, Stanford University, Stanford, CA, 94305
\textsuperscript{b}Department of Aeronautics and Astronautics, Stanford University, Stanford, CA, 94305
\textsuperscript{c}Department of Mechanical Engineering, Stanford University, Stanford, CA, 94305

Abstract

A homogenized flux and body force approach for modeling compressible viscous flows through porous wall is described. The homogeneous model computes the flux through the porous wall as a weighted average of the flux on the wall and the flux through the pore, and takes into account the friction loss on the pore boundary as a body force term. The approach avoids the pore-level resolution mesh, therefore, allows for incorporating porosity for practical use in complex problems, like space landing parachute simulations. Moreover, the proposed model takes account of the compressibility of the flow and does not require the prescribed mass flow rate or discharge coefficient, which marks key differences from other homogenized porous models. To test the homogenized model, a series of pore-level resolved direct numerical simulations with different simple pore geometries and inflow Mach numbers are conducted. The comparisons with these simulations show that the proposed model provides accurate predictions of homogenized quantities near the porous wall.

1. Introduction

Porous walls and membranes appear in a wide range of engineering and scientific applications, such as filtration processes\cite{30,10,24} to separate particulates or molecules from a bulk fluid, multiperforated plates in gas turbines\cite{1,27,26,28} to produce the necessary cooling, parachute system decelerators\cite{20,19,8,15}, and windbreaks or shelterbelts\cite{18,34,40} to reduce the relative flow speed and maintain stability. However, pore-level resolved simulations for most of these large scale engineering applications are still computationally unaffordable. Therefore, high precision and efficient modeling of the fluid flow behavior in such systems is important to improve the understanding of the effects of porous materials and to develop porous wall and membrane related processes and technologies.

The modeling of fluid flows through porous walls or membranes has been inspired by a variety of investigations of flows in porous media. Kim and Peskin\cite{20} and Gao et al.\cite{8} coupled Darcy’s law and Ergun’s law to introduce an internal pressure jump boundary condition into the incompressible Navier–Stokes equations for the simulation of a porous parachute. Wilson\cite{43} and Wang et al.\cite{40} modeled a windbreak similar to a plant canopy by adding a local momentum extraction source term near the windbreak. They calibrated the pressure-loss coefficient in the source term using the pressure drop across the windbreak and the momentum of the flow measured in wind tunnel experiments\cite{18}. Nassehi\cite{30} and Griffiths et al.\cite{10} analyzed cross-flow filtration with the Navier–Stokes equations by applying the slip-wall boundary condition\cite{2} on a porous tube in order to investigate the transport of solutes or particles in a tube with porous walls. Notably, all of the aforementioned investigations are limited to low Mach number (incompressible) flows.

There are only a few research studies regarding the effects of permeability on high Mach number compressible flows. Those studies combined Darcy’s equation with the continuity equation\cite{29}, or derived compressible variants of Darcy’s law\cite{35,37} from the continuity equation and the force balance equation between drag and the pressure drop across an element of porous material. There have also been several efforts that aimed to model the porous

Email addresses: zhengyuh@stanford.edu (Daniel Z. Huang), wongml@stanford.edu (Man Long Wong), lele@stanford.edu (Sanjiva K. Lele), cfarhat@stanford.edu (Charbel Farhat)
surface for film-cooling in gas turbines under high-speed flight conditions. Apte and Yang \cite{Apte2010} treated the porous chamber wall as a mass inflow condition with white noise in the mass flow rate. Mendez and Nicoud \cite{Mendez2018} built a homogeneous model for the multiperforated plate in which the homogenized mass flux is related to the constant discharge coefficient estimated from the pore-level resolved simulations.

The present work proposes a homogenized model for a porous wall boundary in high Mach number compressible viscous flows. To begin, the homogenized fluxes through the porous wall are weighted averages of fluxes through pores and fluxes on the solid wall part. The friction loss due to the pore boundary is modeled as a body force term near the porous wall which depends on the thickness of the porous wall and an assumed pore geometry. Compared with previous models, the present model does not require prescribing the mass flow rate, discharge coefficient, and/or other permeability parameters.

The homogenized porous model is verified using the Mars Science Laboratory parachute fabric and a porous flow model problem at the typical supersonic free-stream Mars landing conditions. High Mach number flows through porous membranes initiated with shocks hitting the porous membranes are simulated with shock Mach numbers ranging between 1.46 and 2.31. Compared with pore-level resolved direct numerical simulation (DNS) results, this homogenized model is able to predict homogenized quantities with reasonable accuracy.

The remainder of the paper is organized as follows. First, the governing equations are introduced in section \ref{sec:gov_eq}. Then, the homogenized porous model suitable for practical simulations is presented in section \ref{sec:hom_model}. The performance of the proposed model is assessed in section \ref{sec:results} by comparing its results with their counterparts from pore-level resolved DNS. Finally, conclusions are offered in section \ref{sec:conclusions}.

2. Governing equations

Throughout this paper, flows are assumed to be compressible and viscous, and are governed by the Navier–Stokes equations, which are written as:

$$\frac{\partial W}{\partial t} + \nabla \cdot F(W) = \nabla \cdot G(W),$$

where $W$ is the vector of the conservative variables describing the fluid state. $F(W)$ and $G(W)$ are respectively the inviscid and viscous flux tensor functions of the conservative variables. Specifically,

$$W = \begin{pmatrix} \rho \\ \rho v \\ E \end{pmatrix}, \quad F(W) = \begin{pmatrix} \rho v \\ \rho v \otimes v + pI \\ (E + p)v \end{pmatrix}, \quad \text{and} \quad G(W) = \begin{pmatrix} 0 \\ \tau \\ \tau \cdot v - q \end{pmatrix},$$

where $\rho$, $v$, and $E$ denote the density, velocity, and total energy per unit volume of the fluid, respectively. They are given by:

$$v = (u, v, w)^T \quad \text{and} \quad E = \rho e + \frac{1}{2} \rho (u^2 + v^2 + w^2),$$

where $e$ denotes the specific (i.e., per unit of mass) internal energy. In the physical inviscid flux tensor $F$, $p$ is the static pressure and $I \in \mathbb{R}^3$ is the identity matrix. In the viscous flux tensor $G$, $\tau$ and $q$ denote the viscous stress tensor and heat flux vector, respectively, and are defined as:

$$\tau = \mu (\nabla v + \nabla v) - \frac{2}{3} \mu (\nabla \cdot v)I \quad \text{and} \quad q = -\kappa \nabla T,$$

where $\mu$ is the dynamic shear viscosity, and $\kappa$ is the thermal conductivity. $T$ is the temperature.

The system of equations eq. \eqref{eq:gov_eq} is closed by assuming that the gas is ideal and calorically perfect:

$$p = \rho RT \quad \text{and} \quad e = \frac{R}{\gamma - 1} T,$$

where $R$ and $\gamma$ are the gas constant and specific heat ratio, respectively.
3. Homogenized flux-body force approach

In this section, the homogenized approach with sub-cell modeling is introduced using a simplified model problem with flow through a porous wall (see figure 1). The porous wall is placed at the center of a square domain, which is periodic and symmetric in both transverse directions (y and z directions). Therefore, periodic boundary conditions can be imposed in both transverse directions by leveraging the periodicity of the porous wall and assuming that the flows are laminar and symmetric.

![Figure 1: Geometry of a three-dimensional porous flow benchmark problem for which: experimental data is only available at low Mach number regime; and DNS-based reference solutions can be computed at both low and high Mach number regimes.](image)

In the homogenized approach, the mesh spacing is assumed to be much larger than the pore size, and the thickness of the porous wall is ignored. A homogenized quantity $\bar{\phi}$, such as a spanwise-averaged or cell-averaged quantity of an original quantity $\phi$, is written as:

$$\bar{\phi} = \frac{\int_{\Omega} \phi d\Omega}{|\Omega|},$$

(6)

where $\Omega$ is the homogenizing area. For the model problem, we take homogenization in both transverse directions with the domain cross-sectional area. The homogenized transverse velocity components $\bar{v}$ and $\bar{w}$ are zero due to the symmetry of the problem setup. The homogenized eq. (1) are given as:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial \rho u}{\partial x} = 0,$$

$$\frac{\partial \rho u}{\partial t} + \frac{\partial (\rho uu + p)}{\partial x} = \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z},$$

$$\frac{\partial E}{\partial t} + \frac{\partial (E + p)u}{\partial x} = \frac{\partial \tau_{xx} u - q_x}{\partial x} + \frac{\partial \tau_{xy} u}{\partial y} + \frac{\partial \tau_{xz} u}{\partial z}. $$

(7)

Specifically, $\tau_{xx}$, $\tau_{xy}$, and $\tau_{xz}$ are components of viscous stress tensor $\tau$ which have expressions as:

$$\tau_{xx} = \mu \left( \frac{4}{3} \frac{\partial u}{\partial x} - \frac{2}{3} \frac{\partial v}{\partial y} - \frac{2}{3} \frac{\partial w}{\partial z} \right), \quad \tau_{xy} = \mu \left( \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right), \quad \tau_{xz} = \mu \left( \frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right),$$

(8)

and $q_x$ is the $x$-component of heat flux vector $q$ that has expression as:

$$q_x = -\kappa \frac{\partial T}{\partial x}.$$ 

(9)

3.1. Homogenized inviscid flux

The inviscid flux derivative on the left hand side of eq. (7) is discretized by the finite volume method. We assume the control volume boundary between node $i$ and node $j$ coincides with the porous wall (see fig. 2). Therefore, inviscid fluxes in the $x$ direction, $F_{ij}^{ave}$, and $F_{ji}^{ave}$, on both sides of the porous wall should be treated specifically. Let $\alpha$ denote the void fraction or porosity of the porous wall. Before constructing the homogeneous fluxes, we consider two extreme cases: when the porosity $\alpha = 1$, the fluxes through the porous wall are regular fluxes, which can be approximated by Riemann solver such as Roe’s solver [36]:

$$F_{ij}^{ave} = -F_{ji}^{ave} = F_{ij}^{Roe}(\mathbf{W}_i, \mathbf{W}_j, n_{ij}),$$

(10)
where \( \mathbf{W}_i \) and \( \mathbf{W}_j \) are homogenized conservative state variables on both sides of the porous wall, and \( n_{ij} \) is the unitary outer normal to the control volume boundary. When the porosity \( \alpha = 0 \), the porous wall degenerates to a solid wall. The fluxes are approximated as wall boundary fluxes:

\[
F_{ij}^{\text{ave}} = F_{ij}^{\text{wall}}(\mathbf{W}_i, n_{ij}) \quad \text{and} \quad F_{ji}^{\text{ave}} = F_{ji}^{\text{wall}}(\mathbf{W}_j, n_{ji}).
\]

Eventually, for any porosity \( \alpha \), the homogenized inviscid fluxes through the porous wall are modeled as weighted averaged fluxes, as follows:

\[
F_{ij}^{\text{ave}} = \alpha F_{ij}^{\text{Roe}}(\mathbf{W}_i, \mathbf{W}_j, n_{ij}) + (1 - \alpha) F_{ij}^{\text{wall}}(\mathbf{W}_i, n_{ij}),
\]

\[
F_{ji}^{\text{ave}} = -\alpha F_{ji}^{\text{Roe}}(\mathbf{W}_i, \mathbf{W}_j, n_{ij}) + (1 - \alpha) F_{ji}^{\text{wall}}(\mathbf{W}_j, n_{ji}).
\]

The cells adjacent to the porous wall are divided into two parts (see fig. 2): the part attached to the hole (colored in blue); the part attached to the wall (colored in red). The flux through the porous wall consists of the flux related to the blue part and the flux associated with the red part, which correspond to the two terms in eq. (12). It is worth mentioning that the homogenized inviscid fluxes conserve both mass and energy and the wall fluxes have zero mass and energy components.

**3.2. Homogenized viscous flux and body force**

The viscous part on the right hand side of eq. (7) is discretized by the finite element method (as same as the 2nd order finite difference method for the one-dimensional model problem). The wall normal viscous flux terms, \( \partial (\tau_{xx} - q_x) / \partial x \), and, \( \partial (\tau_{xx} u - q_x) / \partial x \), are further simplified as follows, due to the periodicity:

\[
\tau_{xx} = \frac{4}{3} \mu \frac{\partial u}{\partial x} \quad \text{and} \quad \tau_{xx} u - q_x = \frac{4}{3} \mu \frac{\partial u}{\partial x} u + \kappa \frac{\partial T}{\partial x}.
\]

The wall normal viscous fluxes in the \( x \) direction, \( G_{ij}^{\text{ave}} \), and \( G_{ji}^{\text{ave}} \), are modeled as weighted averaged fluxes, similar as the inviscid fluxes in section 3.1:

\[
G_{ij}^{\text{ave}} = \alpha \begin{pmatrix}
0 \\
\frac{4}{3} \mu \left( \frac{\pi_j - \pi_i}{\delta_x} \right) \left( \frac{\pi_j + \pi_i}{2} \right) + \kappa \left( \frac{T_j - T_i}{\delta_x} \right)
\end{pmatrix} + (1 - \alpha) \begin{pmatrix}
0 \\
\frac{4}{3} \mu \left( \frac{\pi_j - \pi_i}{\delta_x} \right) \left( \frac{\pi_j + \pi_i}{2} \right) + \kappa \left( \frac{T_j - T_i}{\delta_x} \right)
\end{pmatrix},
\]

\[
G_{ji}^{\text{ave}} = \alpha \begin{pmatrix}
0 \\
\frac{4}{3} \mu \left( \frac{\pi_j - \pi_i}{\delta_x} \right) \left( \frac{\pi_j + \pi_i}{2} \right) + \kappa \left( \frac{T_j - T_i}{\delta_x} \right)
\end{pmatrix} + (1 - \alpha) \begin{pmatrix}
0 \\
\frac{4}{3} \mu \left( \frac{\pi_j - \pi_i}{\delta_x} \right) \left( \frac{\pi_j + \pi_i}{2} \right) + \kappa \left( \frac{T_j - T_i}{\delta_x} \right)
\end{pmatrix},
\]

where \( \pi_i, \pi_j, T_i, \text{ and } T_j \) are ghost velocity and temperature variables, which are populated based on the no-slip wall boundary condition and the adiabatic wall boundary condition, \( \delta_x \) is the grid spacing in the \( x \) direction.

The other terms, \( \partial \tau_{xy} / \partial y + \partial \tau_{xz} / \partial z \), and, \( \partial \tau_{xy} u / \partial y + \partial \tau_{xz} u / \partial z \), vanish away from the pore, since they contain derivatives in the homogenized directions. These terms characterize the friction loss near the pore boundary or at the expanded jet stream boundary, where the solution is not \( C^1 \) continuous. These terms can only be evaluated if fully resolved wall normal velocity component profile \( u \) in the pore is available, thus requires sub-cell modeling.

The proposed model is inspired by the Hagen-Poiseuille flow that is homogeneous in the \( x \) direction and begins by postulating a parabolic shape function for the wall normal velocity component profile, \( u = u(y, z) \), in the pore. The shape function depends on the geometry of the pore. Several pore geometries are considered in the current study including geometries with circular, square, and gapped shapes (see fig. 3).
3.2.1. Circular pore

For the circular geometry shown in fig. 3(a), the velocity profile in the pore is given by:

\[ u(y, z) = C (r^2 - y^2 - z^2), \]

where \( r \) is the radius of the circular pore and \( C \) is a parameter. The averaged velocity satisfies:

\[ \bar{u} = \frac{\int_\Omega u \, dydz}{|\Omega|} = C\alpha r^2. \]

From eq. (15) and eq. (16), the following equations hold:

\[ \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} = -8 \frac{\bar{u}}{r^2}, \]

\[ \frac{u}{\left(\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)^2} = 8 \frac{\pi^2}{\alpha r^2}. \]

(17)

Substituting eq. (17) into eq. (8) leads to the sub-cell model of the friction loss for the circular pore case:

\[ \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} = -8 \mu \frac{\bar{u}}{r^2}, \]

\[ \frac{\partial \tau_{xy} u}{\partial y} + \frac{\partial \tau_{xz} u}{\partial z} = 0. \]

(18)

3.2.2. Square pore

For the square geometry shown in fig. 3(b), the velocity profile in the pore is given by [12, p. 113]:

\[ u(y, z) = C \frac{16r^2}{\pi^3} \sum_{i=1,3,5,...}^{\infty} \left[ (-1)^{(i-1)/2} \left( 1 - \frac{\cosh (i\pi z/2r)}{\cosh (i\pi/2)} \right) \frac{\cos (i\pi y/2r)}{i^3} \right], \]

where \( 2r \) is the width of the square pore, and \( C \) is a parameter. The averaged velocity satisfies:

\[ \bar{u} = \frac{\int_\Omega u \, dydz}{|\Omega|} = C\alpha r^2 \left[ 1 - \frac{192}{\pi^5} \sum_{i=1,3,5,...}^{\infty} \frac{\tanh (i\pi/2)}{i^5} \right]. \]

(20)

From eq. (19) and eq. (20), the following equations hold:

\[ \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} = -\frac{3 \pi \bar{u}}{r^2}, \]

\[ \frac{u}{\left(\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right)^2} + \left(\frac{\partial u}{\partial y}\right)^2 + \left(\frac{\partial u}{\partial z}\right)^2 = 0. \]

(21)
where
\[ \zeta = 1 - \frac{192}{\pi^5} \sum_{i=1,3,5,\ldots}^{\infty} \frac{\tanh(i\pi/2)}{i^5} \approx 0.421731044865. \]

Substituting eq. (21) into eq. (8), the sub-cell model of the friction loss for the square pore case is given by:
\[ \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} = \frac{3}{\zeta \mu} \frac{\pi}{r^2}, \]
\[ \frac{\partial \tau_{xy} u}{\partial y} + \frac{\partial \tau_{xz} u}{\partial z} = 0. \]  

(22)

### 3.2.3. Gapped shape pore

For the gapped shape shown in fig. 3(c) the velocity profile in the pore is given by:
\[ u(y, z) = C (r^2 - y^2), \]  

(23)

where \( 2r \) is the width of the gap and \( C \) is a parameter. The averaged velocity satisfies:
\[ \bar{u} = \frac{\int_D u \, dy dz}{|D|} = C \frac{2\alpha r^2}{3}. \]  

(24)

From eq. (23) and eq. (24), the following equations hold:
\[ \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} = -3 \frac{\pi}{r^2}, \]
\[ u \left( \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) = -3 \frac{\pi^2}{\alpha r^2}, \]
\[ \left( \frac{\partial u}{\partial y} \right)^2 + \left( \frac{\partial u}{\partial z} \right)^2 = 3 \frac{\pi^2}{\alpha r^2}. \]  

(25)

Substituting eq. (25) into eq. (8) leads to the following sub-cell model of the friction loss for the gap:
\[ \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} = -3\mu \frac{\pi}{r^2}, \]
\[ \frac{\partial \tau_{xy} u}{\partial y} + \frac{\partial \tau_{xz} u}{\partial z} = 0. \]  

(26)

All scenarios reveal that the friction loss contributes only to the momentum loss, but not the total energy, since kinetic energy is degraded to internal energy by the friction. For certain void fraction, e.g. \( \alpha = 8\% \), the momentum loss for a circular pore in eq. (18) is about 314.16(\( \mu u \)), the momentum loss for a square pore in eq. (22) is about 348.56(\( \mu u \)), and the momentum loss for a gap in eq. (26) is about 1875(\( \mu u \)). This explains that the smaller aspect ratio holes lead to smaller momentum loss or dissipation.

These pore shape dependent source or body force terms given by eqs. (18), (22) and (26) are added near the porous wall by using a Dirac delta function, which is approximated by the Gaussian function:
\[ D(x) = \exp \left[ -\frac{1}{2} \left( \frac{x - x_0}{\sigma} \right)^2 \right], \]  

(27)

where \( x_0 \) is the porous wall position, and \( \sigma = \eta/(2\pi) \) where \( \eta \) is the thickness of the porous wall or membrane. Note that the integral of \( D(x) \) equals to the membrane thickness. The friction loss is incorporated into the model as a source term \( S \) given by:
\[ S = \left( L_m D(x) \left( \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right) \right), \]  

(28)

where \( L_m \) represents the effective membrane thickness factor which is needed, because besides the pore boundary, the expanded jet stream through the pore (\( C^0 \) continuous solution) also contributes to the momentum loss.
4. Verification and application problems

In this section, the homogenized porous model is verified on the Mars Science Laboratory parachute fabric, the PIA-C-7020 Type I fabric with Ripstop weave, under typical supersonic free-stream Mars landing conditions. The porous fabric [4] (see fig. 4) has about 8% porosity based on the X-ray microtomography result in [33], the thickness is around 80 µm [23], and the periodicity is about 500 µm in both directions. Due to the lack of supersonic permeability test data on this material, pore-level resolved DNS results are used as reference solutions. Moreover, as discussed in section 3.2, the pore geometry has significant impact on the friction loss. Hence, the equivalent pore geometry of the complicated Ripstop weave pattern is first explored based on low Mach number permeability test data and the DNS results in section 4.2.

4.1. Problem setup

To setup the simulations, several simplifications are made here. Firstly, by leveraging the periodicity of the weave pattern, we simulate only a small square fabric piece of length 500 µm, which corresponds to the weave repeat size. Secondly, the fabric is assumed to be rigid, and the weave pattern is simplified by lumping all slots between spun fibres into a center rectangular hole, which matches the void fraction $\alpha$. The simplified geometries are shown in fig. 5 for various pore aspect ratios — 1:1, 2:1, 8:1, and 12.5:1. The piece of fabric is placed at the center of the computational domain (see fig. 1). Periodic boundary conditions are imposed in both transverse directions and different initial conditions and inflow/outflow boundary conditions are applied.

![Simplified geometries](image)

Figure 5: Simplified symmetric geometries of the fabric piece with around 8% void fraction. The center rectangular holes are of various aspect ratios: (a) 1:1 (b) 2:1 (c) 8:1 (d) 12.5:1.

4.2. Explore equivalent pore geometry

The permeability of the aforementioned parachute fabric was measured by Cruz et al. [4]. The test was conducted using a Textest Instruments FX 3300 Labotester III Air Permeability Tester, which consists of a test chamber with
Figure 6: The Mach number profiles in central $x$-$y$ plane for cases with holes of different aspect ratios: $2/7 \times 2/7$ (top-left), $20\% \times 40\%$ (top-right), $10\% \times 80\%$ (bottom-left), and $8\% \times 100\%$ (bottom-right), with an initial pressure difference of 3000 Pa, at $t = 15.8 \mu s$.

a vacuum pump. The sample fabric was clamped over the test head opening, therefore the upstream conditions were at the atmospheric pressure. The differential pressure was selected and maintained by the vacuum pump in the chamber and the resultant permeability in terms of flow velocity was measured.

Modeling the entire chamber and pumping process is computationally unaffordable and mathematically challenging, thus the pore-level resolved DNS are carefully designed as following, to mimic the experimental conditions [4].

The computational domain is of size $4 \text{ mm} \times 0.5 \text{ mm} \times 0.5 \text{ mm}$ in the $x$, $y$, and $z$ directions. Extrapolation boundary conditions are enforced at the left and right boundaries in $x$ direction so as to minimize reflection of spurious waves back into the computational domain. The gas properties of the air are taken as:

$$\gamma = 1.4, \quad R = 287.2 \text{ J kg}^{-1} \text{K}^{-1}, \quad \text{and} \quad \mu = 1.846 \times 10^{-5} \text{ kg m}^{-1} \text{s}^{-1}. \quad (29)$$

At the initial time, the quiescent air in the computational domain is uniformly at the laboratory’s temperature $T = 296.5 \text{ K}$ and the pressure upstream is at the laboratory’s atmospheric pressure $p = 94625 \text{ Pa}$. The pressure downstream is chosen to form an initial pressure difference, which spans a wide range (100 Pa, 200 Pa, 500 Pa, 1000 Pa, 2000 Pa, 3000 Pa, and 4000 Pa). It is worth mentioning that these initial pressure differences are not the pressure differences in the experimental results.

At the beginning of each simulation, pressure waves propagate in both directions away from the porous membrane. Meanwhile, the pressure difference across the porous wall drops, the fluid flow near the porous wall accelerates immediately. A weak jet stream forms downstream, which is depicted in fig. by the Mach number profile in the central $x$-$y$ plane at time $t = 15.8 \mu s$ for each case. There are major differences in the Mach number profiles among different aspect ratio holes. The smaller aspect ratio holes ($2/7 \times 2/7$ and $20\% \times 40\%$) lead to larger maximum Mach numbers at around 0.2, which verifies our explanation about the friction loss in section 3.2.

The pressure difference across the porous membrane tends to become quasi-steady at late times and the velocity on both sides of porous membrane also converges at about $t = 15.8 \mu s$ for each case. Fig. compares the relation between the pressure difference and the (permeability) flow velocity measured in [4] with 99.7 confidential interval and the relations obtained in the present simulations with different pore geometries in fig. In the present simulations, the pressure difference is evaluated as the pressure jump across the fabric at $t = 15.8 \mu s$, when the pressure difference tends to stabilize. The permeability velocity through the porous membrane, $u_{\text{permeability}}$, is evaluated as the averaged mass flux on any cross section, $\Omega$, divided by the initial upstream density, $\rho_{\text{upstream}}$:

$$u_{\text{permeability}} = \frac{\int_{\Omega} \rho u d\Omega}{\rho_{\text{upstream}}}.$$  \quad (30)
Figure 7: A comparison of the relations between velocity and differential pressure from the experimental measurements and DNS results. The experimental measurements are denoted by blue circles with 99.7 confidential interval error bars. The DNS results are denoted by other symbols. Orange circles: 2/7 × 2/7 aspect ratio hole; green squares: 20% × 40% aspect ratio hole; red diamonds: 10% × 80% aspect ratio hole; purple triangles: 8% × 100% aspect ratio hole.

although, in such an incompressible regime the density fluctuation is negligible at roughly 2%. The agreement of these numerical results and the experimental results is good, though there are deviations for small aspect ratio pore geometries. These deviations may be due to the fact that these slots in the fabric are aggregated into solo holes. According to [6], for a given permeability, the total pressure drop, \((\Delta p)^{\text{total}}\), can be split into components due to the viscous \((\Delta p)^{\text{viscous}}\) and inertial \((\Delta p)^{\text{inertial}}\) effects:

\[
(\Delta p)^{\text{total}} = (\Delta p)^{\text{viscous}} + (\Delta p)^{\text{inertial}}.
\] (31)

At high Reynolds number, the inertial effect is dominant, whereas the viscous effect prevails at low Reynolds number. Our current study falls into the latter category. In this category, one contribution of the viscous effect is the friction loss due to the shear stress near the surface of each spun fibre. And pore geometries with smaller aspect ratio holes (2/7 × 2/7 and 20% × 40%) corresponding with smaller pore surface, therefore, leads to smaller pressure difference for a given velocity through the porous membrane. Therefore, the geometries with large aspect ratio holes (10% × 80% and 8% × 100%) are closer to the experimental results of the Mars Science Laboratory parachute fabric.

4.3. Model verification under Mars landing conditions

There is a lack of research regarding the effects of permeability on high Mach number compressible flows, especially when there are interactions between shock waves and porous membranes. This is significant to understand the dynamics of supersonic parachute decelerators [3, 20, 15, 35]. The DNS results of shock interacting with the porous wall are reported in this section. Specifically, planar shock waves at different Mach numbers initialized 250 \(\mu\)m ahead of the porous membrane (see fig. 8), compressing the flow, hitting the porous wall and rebounding back, are simulated.

The initial post-shock conditions are set as the Mars landing conditions — the low density and low pressure conditions:

\[
p = 260 \text{ Pa} \quad \text{and} \quad \rho = 0.0067 \text{ kg m}^{-3}.
\] (32)

The Mars atmosphere mainly consists of carbon dioxide, CO\(_2\). The gas properties are given by:

\[
\gamma = 1.33, \quad R = 188.4 \text{ J kg}^{-1}\text{K}^{-1}, \quad \text{and} \quad \mu = 1.03 \times 10^{-5} \text{ kg m}^{-1}\text{s}^{-1}.
\] (33)

Note that the bulk viscosity effect is not considered in the simulations but there is a brief discussion on the effect in appendix C. The initial shock Mach numbers, \(M_s\), considered are 1.46, 1.96, and 2.31. In all cases, the flows
in the pre-shock regime are quiescent and the initial conditions are obtained from Rankine–Hugoniot shock jump conditions. The detailed initial conditions are listed in tab. 1.

| $M_s$ | Post-shock regime                                                                 | Pre-shock regime                                                                 |
|-------|-----------------------------------------------------------------------------------|-----------------------------------------------------------------------------------|
| 1.46  | $\rho = 0.0067 \text{ kg m}^{-3}, u = 127.98 \text{ m s}^{-1}, p = 260 \text{ Pa}$ | $\rho = 0.004119 \text{ kg m}^{-3}, u = 0.0 \text{ m s}^{-1}, p = 112.73 \text{ Pa}$ |
| 1.96  | $\rho = 0.0067 \text{ kg m}^{-3}, u = 213.31 \text{ m s}^{-1}, p = 260 \text{ Pa}$ | $\rho = 0.002774 \text{ kg m}^{-3}, u = 0.0 \text{ m s}^{-1}, p = 61.240 \text{ Pa}$ |
| 2.31  | $\rho = 0.0067 \text{ kg m}^{-3}, u = 255.97 \text{ m s}^{-1}, p = 260 \text{ Pa}$ | $\rho = 0.002300 \text{ kg m}^{-3}, u = 0.0 \text{ m s}^{-1}, p = 43.772 \text{ Pa}$ |

Table 1: Initial conditions of post-shock and pre-shock regimes in the simulations.

The simplified porous membrane geometries (see fig. 5) with $2/7 \times 2/7$ and $8\% \times 100\%$ (a gap) aspect ratio holes are chosen in this section. The computational domain is of size $10 \text{ mm} \times 0.5 \text{ mm} \times 0.5 \text{ mm}$ in the $x$, $y$, and $z$ directions. Extrapolation boundary conditions are enforced at the left and right boundaries in $x$ direction so as to minimize spurious reflection of shock waves, and periodic boundary conditions are used in both transverse directions.

In each case, when the shock bounces back from the porous wall, a high density and high pressure zone is generated upstream of the porous wall. As a result, the flow through the small hole is like flow through a converging-diverging nozzle. Downstream of the porous membrane, the flow immediately accelerates and expands to low density and low pressure conditions. Fig. 9 shows the Mach number profiles at the end time $t = 23.46 \mu s$. As we can see, it is similar with the low Mach number scenario (see fig. 6) that small aspect ratio holes lead to jet streams at higher Mach numbers. However, we also observe that flows are choked for all these high Mach number cases. Fig. 10 depicts the streamlines on the $x$-$y$ plane for each case. Note that we can see that the flow expands downstream of the porous membrane and pushes the circulations to the corner for the two highest shock Mach number cases with high aspect ratio hole.

The Reynolds numbers based on jet flow velocity at the pore exit are reported in tab. 2. The Reynolds number $Re$ is defined as:

$$Re = \left( \frac{\int_A \rho u^2 dA}{\rho_{\text{upstream}} d} \right)^{1/2} \left( \frac{\rho_{\text{upstream}} d}{\mu} \right),$$

(34)

where $A$ is the pore area and $d = \sqrt{4A/\pi}$ is the equivalent pore diameter. These Reynolds numbers are very low due to the low density and the small pore size, which illustrate the significance of modeling the viscous effect.

| Aspect ratio of hole | $M_s = 1.46$ | $M_s = 1.96$ | $M_s = 2.31$ |
|---------------------|--------------|--------------|--------------|
| $2/7 \times 2/7$    | 7.3          | 13.50        | 17.96        |
| $8\% \times 100\%$ | 16.43        | 23.75        | 27.38        |

Table 2: Jet flow velocity based Reynolds numbers, $Re$, for simulations with different aspect ratio holes and initial shock Mach numbers, $M_s$.

Furthermore, the averaged mass flow rate is plotted against the pressure difference on both sides of the porous membrane for each aspect ratio case in fig. 11. The data set shows an approximate linear relation given by:

$$\dot{m} = K \Delta p,$$

(35)
Through least square fitting, for the cases with $2/7 \times 2/7$ aspect ratio hole, the coefficient is found as $K = 1.61 \times 10^{-4}$ s m$^{-1}$, and for the cases with $8\% \times 100\%$ aspect ratio hole, the coefficient is found as $K = 8.54 \times 10^{-5}$ s m$^{-1}$. The linear relation satisfies the Darcy’s law and the coefficient depends only on the center pore geometry.

The homogenized model is applied to the aforementioned high Mach number compressible flow through porous membrane cases. The one dimensional computational domain (due to homogenization in both transverse directions) for the model simulations is $x \in [-0.5, 0.5]$ cm. Finite volume and finite element methods are used respectively for the inviscid and viscous fluxes (see appendix A). The time discretization is performed using the second-order explicit Runge–Kutta scheme with a CFL number of 0.5. All three sets of simulations of $M_s = 1.46, 1.96,$ and $2.31$ shock waves interacting with the porous membranes with different aspect ratio holes are conducted using the present porous model with 400 and 2000 uniform grid points. For the cases with $2/7 \times 2/7$ aspect ratio hole, the jet stream expands immediately downstream the membrane. Hence, the effective membrane thickness factor $L_m$ is set to be 1; For the cases with $8\% \times 100\%$ aspect ratio hole, the jet stream remains longer, thus the effective membrane thickness factor $L_m$ is set to be 4. In figs. 12 to 14, it is shown that the predictions of averaged density, streamwise velocity, and pressure profiles by the present homogenized model at $t = 23.46$ µs reasonably agree with the pore-level resolved DNS results. It is worth mentioning that the averaged density and pressure are much less sensitive to the factor $L_m$ compared to the averaged velocity.

Moreover, the predicted drag forces are depicted in fig. 15. For the pore-level resolved DNS, the net drag force consists of forces due to pressure jump $\Delta p$ and the normal shear stress jump $\Delta \tau_{xx}$ across the membrane, and the shear forces on the pore boundaries. We observe that the net shear force on the pore boundaries roughly equal to the net “virtual” force due to pressure and normal shear stress differences between the two ends of the pore on the pore cross section. Therefore, in our homogenized model, the drag force $F_D$ is computed as follows:

$$F_D = (\Delta p - \Delta \tau_{xx})S,$$  \hspace{1cm} (36)

where $S$ is fabric area that includes both the solid and void parts. The “virtual” force on the void part approximately captures the net shear force on the pore boundaries and the results match well with the DNS results.
Figure 10: The streamlines in x-y plane for cases with holes of different aspect ratios: 2/7 × 2/7 (left column), 8/100 × 100/100 (right column), and different initial shock Mach numbers: $M_s = 1.46$, 1.96, and 2.31 (top to bottom), at 23.4 μs.

5. Conclusion

A novel homogenized porous model specifically for high Mach number compressible flows through porous membranes is built. Comparing with previous models, which require mass flow rate, discharge coefficients or other permeability parameters, the present homogenized model starts from the first principle, by modifying the convective and diffusive fluxes to take account of the effects of void fraction, and considers the friction loss due to hole geometry through a body force term. The model is valid for different hole geometries, and for various high Mach number flow conditions. The results of the model were found to be in good agreement with the pore-level resolved DNS results. The incorporation of the present model to any numerical software is straightforward (see appendix A). Therefore, the present model enables considering porous effects in practical applications, such as the simulation of parachute inflation process [20, 8, 15].

Moreover, the permeability of the Mars Science Laboratory parachute fabric, the PIA-C-7020 Type I fabric with Ripstop weave, was studied numerically in this work. The equivalent simple pore geometry is built, which matches the complicated Ripstop weave pattern, in terms of the relation between the pressure jump and the velocity obtained from experiments. And in the Mars landing condition simulations, the linear relation between the pressure difference and mass flux was discovered, which conforms to Darcy’s law. However, the Mars atmosphere conditions are simplified, as the bulk viscosity is not incorporated in the present study, while further exploration will be considered. In addition, the fluid structure interaction of the flexible membrane would also be of considerable interest.

Acknowledgments

Daniel Z. Huang and Charbel Farhat acknowledge partial support by the Jet Propulsion Laboratory (JPL) under Contract JPL-RSA No. 1590208, and partial support by the National Aeronautics and Space Administration (NASA) under Early Stage Innovations (ESI) Grant NASA-NNX17AD02G.
Figure 11: Linear fits of averaged mass flux against pressure jump using data collected from the high Mach number pore-level resolved DNS at different times.

Figure 12: Predicted averaged density, streamwise velocity, and pressure profiles (top to bottom) with initial shock Mach number $M_s = 1.46$ and holes of different aspect ratios: $2/7 \times 2/7$ (left) and $8\% \times 100\%$ (right) at $t = 23.46 \mu s$. Blue solid line: pore-level resolved simulation; orange dashed line: present model with 400 grid points; green dash-dotted line: present model with 2000 grid points.
Figure 13: Predicted averaged density, streamwise velocity, and pressure profiles (top to bottom) with initial shock Mach number $M_s = 1.96$ and holes of different aspect ratios: $2/7 \times 2/7$ (left) and $8\% \times 100\%$ (right) at $t = 23.46\ \mu s$. Blue solid line: pore-level resolved simulation; orange dashed line: present model with 400 grid points; green dash-dotted line: present model with 2000 grid points.

Figure 14: Predicted averaged density, streamwise velocity, and pressure profiles (top to bottom) with initial shock Mach number $M_s = 2.31$ and holes of different aspect ratios: $2/7 \times 2/7$ (left) and $8\% \times 100\%$ (right) at $t = 23.46\ \mu s$. Blue solid line: pore-level resolved simulation; orange dashed line: present model with 400 grid points; green dash-dotted line: present model with 2000 grid points.
Appendices

A. Implementation details of the homogenized approach

The implementation details of the homogenized porous wall model in the in-house computational framework with the FInite Volume method with Exact two-material Riemann problems (FIVER) [41, 21, 25, 13, 3] is discussed in this section.

FIVER semi-discretizes the governing eq. 1 by a vertex-based finite volume method for inviscid fluxes and a finite element method for viscous fluxes. Using the standard characteristic function associated with each control volume $C_i$, the standard piecewise linear test function $\psi_i$ associated with each node $i$, and the equivalence between the two functional spaces [7], the weak and semi-discrete form of eq. 1 reads as follows:

$$ \parallel C_i \parallel \frac{\partial W_i}{\partial t} + \sum_{j \in K(i)} \int_{\partial C_{ij}} F(W_h) \cdot n_{ij} \, d\partial \Omega + \int_{\partial C_i \cap \partial \Omega_{\infty}} F(W_h) \cdot n_\infty \, d\partial \Omega + \sum_{e \ni i} \int_{\Omega_e} \nabla \psi^\ast_i \cdot G(W_h) \, d\Omega_e = 0. \quad (37) $$

Here, $\parallel C_i \parallel$ denotes the volume of $C_i$, $\partial C_{ij}$ denotes the cell boundary between $C_i$ and $C_j$, $n_{ij}$ denotes the unit outward normal to $\partial C_{ij}$, $W_h$ denotes the spatial approximation of the fluid state $W$, $W_i$ denotes the average values of $W_h$ in $C_i$, $K(i)$ denotes the set of nodes connected by an edge to the node $i$, $\partial \Omega_{\infty}$ denotes the far-field boundary of the fluid domain with unit outward normal $n_\infty$, and $\psi^\ast_i$ denotes the restriction of $\psi_i$ to the fluid primal cell $\Omega_e$.

At the fluid-structure interface, assuming edge $i-j$ intersects the interface and node $i$ is in the fluid domain as in fig. 16, the convective flux through $\partial C_{ij}$ is evaluated as:

$$ \int_{\partial C_{ij}} F(W_h) \cdot n_{ij} \, d\partial \Omega \approx \parallel \partial C_{ij} \parallel F^{Roe}_{ij}(W_i, W_i^\ast, n_{ij}, EOS_i), \quad (38) $$

which corresponds to $F^{wall}(W_i, n_{ij})$ in the homogenized inviscid fluxes eq. 12. FIVER assumes the midpoint $M$ of the edge $i-j$ coincides with the embedded discrete interface and reconstructs the fluid state vector $W_i^\ast$ at the embedded discrete interface by solving a one-dimensional exact Riemann problem or piston problem [41]. This procedure avoids traversing the fluid-structure interface as in the ghost fluid method. Moreover, solving the half Riemann problem guarantees the characteristic theory and captures well the shock wave or rarefaction wave near
the fluid-structure interface. Reconstruction procedure near the fluid-structure interface is incorporated in \cite{25} to improve the order of the accuracy near the interface.

As for the viscous integral in eq. (37), ghost node procedure is applied to impose boundary conditions in the elements intersected by the fluid-structure interface, such as element $\Omega_{e_{ijk}}$ in fig. 16. The idea here is to populate the velocity and temperature at the inactive node $j$ by a linear or constant extrapolation procedure \cite{21}. Consequently, variables in the primal cell $\Omega_e$ required to evaluate the viscous fluxes can be rewritten as:

$$
V^\epsilon = \frac{1}{N^n_e} \left( \sum_{k=1}^{N^n_a} V_k + \sum_{k=1}^{N^n_a} V_k^g \right) \quad \text{and} \quad \nabla V^\epsilon = \sum_{k=1}^{N^n_a} \nabla \psi^e_k V_k + \sum_{k=1}^{N^n_a} \nabla \psi^e_k V_k^g.
$$

Here $V_k$ denotes the primitive variable vector at the node $k$, including velocity and temperature. $V_k^g$ is the populated ghost variables at node $k$, and $V^\epsilon$ and $\nabla V^\epsilon$ are the averaged variables in the primal cell $\Omega_e$. The viscous flux corresponds to the wall bounded flux in the homogenized viscous fluxes in eq. (14).

As for the body force term, it is added near the porous wall. The approximated delta function $D(x)$ (see eq. (27)) is evaluated based on the distance to the porous wall. For efficiency purpose, adding the source term to one or two layers of mesh away from the porous wall is enough.

### B. Details of the pore-level resolved DNS

The pore-level resolved DNS are performed using the in-house Hydrodynamics Adaptive Mesh Refinement Simulator (HAMeRS) \cite{47} \cite{46} \cite{45}. The parallelization of the code and all construction, management and storage of adaptive cells, the communication at coarse-fine patch interfaces, and the multi-time stepping, are facilitated by the Structured Adaptive Mesh Refinement Application Infrastructure (SAMRAI) library \cite{44} \cite{11} \cite{12} from Lawrence Livermore National Laboratory (LLNL).

The derivative of the inviscid flux vector in eq. (1) is semi-discretized on a uniform grid by a high-order accurate nonlinear finite difference scheme that belongs to the family of Weighted Compact Nonlinear Schemes (WCNS)

Figure 16: Spatial discretization of a two-dimensional fluid computational domain and a structure. Active and inactive nodes are marked, respectively, with black and white bullets. The exact two-material Riemann problem is at the surrogate material interface $\partial C_{ij}$. 

Figure 16: Spatial discretization of a two-dimensional fluid computational domain and a structure. Active and inactive nodes are marked, respectively, with black and white bullets. The exact two-material Riemann problem is at the surrogate material interface $\partial C_{ij}$. 

Figure 16: Spatial discretization of a two-dimensional fluid computational domain and a structure. Active and inactive nodes are marked, respectively, with black and white bullets. The exact two-material Riemann problem is at the surrogate material interface $\partial C_{ij}$.
originally proposed by Deng and Zhang [5]. For the approximation of the derivative of the inviscid flux in the x direction $F$, the sixth order accurate Midpoint-and-Node-to-node Differencing (MND) scheme proposed by Nonomura [31] is used:

$$\frac{\partial \tilde{F}}{\partial x_{i,j,k}} = \frac{1}{\delta x} \left[ \frac{3}{2} \left( \tilde{F}_{i+\frac{1}{2},j,k} - \tilde{F}_{i-\frac{1}{2},j,k} \right) - \frac{3}{10} \left( F_{i+1,j,k} - F_{i-1,j,k} \right) + \frac{1}{30} \left( \tilde{F}_{i+\frac{3}{2},j,k} - \tilde{F}_{i-\frac{3}{2},j,k} \right) \right],$$  

where $\delta x$ is the grid spacing in the x direction. The scheme approximates the flux derivative using the flux vector $\tilde{F}$ at the midpoint between cell nodes approximated with a fifth order accurate upwind-biased Weighted Essentially Non-Oscillatory (WENO) interpolation originated from the nonlinear weighting technique by Jiang and Shu [16]. The characteristic variables projected from primitive variables are interpolated, which is observed to suppress pressure oscillations at contact discontinuities in [17, 32]. The flux can then be approximated by HLLC Riemann solver [39]. However, it is well known that HLLC Riemann solver can give rise to numerical instability near shocks in some multi-dimensional problems when the shock normal direction does not align well with the grid normal surface direction. Therefore, a more robust hybridized HLLC-HLL Riemann solver is used instead in the shock wave region, detected by a Ducros-like shock sensor, $s$, by Larsson et al. [22] under the criteria:

$$s = \frac{-\theta}{|\theta| + |\omega| + \epsilon} > 0.65,$$

where $\theta = \nabla \cdot v$ is the volumetric dilatation rate and $\omega = \nabla \times v$ is the vorticity. $\epsilon$ is a very small value close to machine epsilon to prevent division by zero. The detailed implementation of the hybrid HLLC-HLL Riemann solver can be found in work by Huang et al. [14]. The discretization of the inviscid fluxes in y and z directions is similar with grid spacings $\delta y$ and $\delta z$ respectively.

The derivative of the viscous flux vector $G$ in x direction is spatially approximated by the sixth order central finite difference method:

$$\frac{\partial \tilde{G}}{\partial x_{i,j,k}} = \frac{1}{\delta x} \left[ \frac{1}{60} \left( G_{i+3,j,k} - G_{i-3,j,k} \right) - \frac{3}{20} \left( G_{i+2,j,k} - G_{i-2,j,k} \right) + \frac{3}{4} \left( G_{i+1,j,k} - G_{i-1,j,k} \right) \right],$$

The gradients of velocity and temperature at cell nodes are also computed as eq. [42]. This means conservative discretizations of the viscous fluxes in the Navier–Stokes equations are enforced as eq. [42] is applied repeatedly. The adiabatic wall boundary condition is applied on the rigid fabric wall by the ghost node method. The ghost node values and first order derivatives are populated by mirroring. The third order total variation diminishing Runge–Kutta scheme [9] is used for temporal discretization.

The grid sizes of all simulations presented in current work are listed in tab. 3. All grids are uniform in each directions. Two convergent tests are conducted, including the low Mach number flow through a porous membrane with a 8% × 100% aspect ratio hole with an initial pressure jump of 4000 Pa, and the high Mach number flow induced by a $M_s = 1.46$ shock through a porous membrane with a 2/7 × 2/7 aspect ratio hole. The finer grids have twice number of grid points in each direction. The pressure profiles for both fine and coarse grids are reported in fig. [17] which indicates grid convergence of our current study.

| Problem description                  | Pore size | Domain size | Grid size   |
|-------------------------------------|-----------|-------------|-------------|
| Permeability test                   | 2/7 × 2/7 | 4 mm × 0.5 mm × 0.5 mm | 800 × 182 × 182 |
| Permeability test                   | 20% × 40% | 4 mm × 0.5 mm × 0.5 mm | 800 × 200 × 200 |
| Permeability test                   | 10% × 80% | 4 mm × 0.5 mm × 0.5 mm | 800 × 200 × 200 |
| Permeability test                   | 8% × 100% | 4 mm × 0.5 mm | 1600 × 200 |
| Shock porous wall interaction       | 2/7 × 2/7 | 10 mm × 0.5 mm × 0.5 mm | 2000 × 84 × 84 |
| Shock porous wall interaction       | 8% × 100% | 10 mm × 0.5 mm | 2000 × 200 |

Table 3: Grid information of different simulations.
C. Discussion about the role of bulk viscosity

A more general form of the stress tensor for a compressible Newtonian fluid is given by:

\[ \tau = \mu (\nabla v + \nabla^T v) - \left( \frac{2}{3} \mu - \mu_v \right) \nabla \cdot v I, \]  

(43)

where \( \mu_v \) is the bulk viscosity. However, the bulk viscosity has not been considered in the verification. But it might be interesting to analyze its effect, since the Mars atmosphere mainly consists of CO\(_2\), which is known for its large bulk viscosity. The volumetric dilatation rate, \( \nabla \cdot v \), and the shear strain rate component, \( \partial u/\partial y + \partial v/\partial x \), are depicted in fig. 18 for the case initially with a \( M_s = 1.96 \) shock wave. The volumetric dilatation rate is large near the pore outlet, due to the expansion of the high density and high pressure flow. The homogenized dilatation rate \( \nabla \cdot v \) is approximated by eq. (14) in the present model. The shear strain rate is significant near the pore boundary, which is modeled in section 3.2. Fig. 18 also shows that the shear strain rate is only several times larger than the volumetric dilatation rate. Therefore, based on the present study, the bulk viscosity has a potentially important role under the flow conditions on Mars, since \( \mu_v/\mu \gg 1 \) at the conditions.
Figure 18: Profiles of volumetric dilatation rate (s\(^{-1}\)) (top row) and shear strain rate component (s\(^{-1}\)) (bottom row), \(\partial u/\partial y + \partial v/\partial x\), post-processed from DNS results for high Mach number flows induced by a \(M_s = 1.46\) shock through a 2/7 × 2/7 aspect ratio hole (left) and a 8% × 100% aspect ratio hole (right).
References

[1] Sourabh Apte and Vigor Yang. Unsteady flow evolution in porous chamber with surface mass injection, part 1: Free oscillation. *AIAA journal*, 39(8):1577–1586, 2001.

[2] Gordon S Beavers and Daniel D Joseph. Boundary conditions at a naturally permeable wall. *Journal of fluid mechanics*, 30(1):197–207, 1967.

[3] Raunak Borker, Daniel Huang, Sebastian Grimberg, Charbel Farhat, Philip Avery, and Jason Rabinovitch. Mesh adaptation framework for embedded boundary methods for computational fluid dynamics and fluid-structure interaction. *International Journal for Numerical Methods in Fluids*, 2019.

[4] Juan R Cruz, Clara O’Farrell, Elsa Hennings, and Paul Runnels. Permeability of two parachute fabric—measurements, modeling, and application. In 24th *AIAA Aerodynamic Decelerator Systems Technology Conference*, page 3725, 2017.

[5] Xiaogang Deng and Hanxin Zhang. Developing high-order weighted compact nonlinear schemes. *Journal of Computational Physics*, 165(1):22–44, 2000.

[6] Sabri Ergun and Ao Ao Orning. Fluid flow through randomly packed columns and fluidized beds. *Industrial & Engineering Chemistry*, 41(6):1179–1184, 1949.

[7] Charbel Farhat, Loula Fezoui, and Stéphane Lanteri. Two-dimensional viscous flow computations on the connecti on machine: Unstructured meshes, upwind schemes and massively parallel computations. *Computer Methods in Applied Mechanics and Engineering*, 102(1):61–88, 1993. doi: https://doi.org/10.1016/0045-7825(93)90141-J. URL http://www.sciencedirect.com/science/article/pii/004578259390141J.

[8] Zheng Gao, Richard D Charles, and Xiaolin Li. Numerical modeling of flow through porous fabric surface in parachute simulation. *AIAA Journal*, 55(2):686–690, 2016.

[9] Sigal Gottlieb and Chi-Wang Shu. Total variation diminishing Runge–Kutta schemes. *Mathematics of computation of the American Mathematical Society*, 67(221):73–85, 1998.

[10] IM Griffiths, PD Howell, and RJ Shipley. Control and optimization of solute transport in a thin porous tube. *Physics of Fluids*, 25(3):033101, 2013.

[11] Richard D Hornung and Scott R Kohn. Managing application complexity in the SAMRAI object-oriented framework. *Concurrency and computation: practice and experience*, 14(5):347–368, 2002.

[12] Richard D Hornung, Andrew M Wissink, and Scott R Kohn. Managing complex data and geometry in parallel structured amr applications. *Engineering with Computers*, 22(3-4):181–195, 2006.

[13] Daniel Z Huang, Dante De Santis, and Charbel Farhat. A family of position-and orientation-independent embedded boundary methods for viscous flow and fluid–structure interaction problems. *Journal of Computational Physics*, 365:74–104, 2018.

[14] Kuibang Huang, Hao Wu, Heng Yu, and Dong Yan. Cures for numerical shock instability in HLLC solver. *International journal for numerical methods in fluids*, 65(9):1026–1038, 2011.

[15] Zhengyu Huang, Philip Avery, Charbel Farhat, Jason Rabinovitch, Armen Derkevorkian, and Lee D Peterson. Simulation of parachute inflation dynamics using an Eulerian computational framework for fluid-structure interfaces evolving in high-speed turbulent flows. In *2018 AIAA Aerospace Sciences Meeting*, page 1540, 2018.

[16] Guang-Shan Jiang and Chi-Wang Shu. Efficient implementation of weighted ENO schemes. *Journal of computational physics*, 126(1):202–228, 1996.

[17] Eric Johnsen and Tim Colonius. Implementation of WENO schemes in compressible multicomponent flow problems. *Journal of Computational Physics*, 219(2):715–732, 2006.

[18] MJ Judd, MR Raupach, and JJ Finnigan. A wind tunnel study of turbulent flow around single and multiple windbreaks, part i: velocity fields. *Boundary-Layer Meteorology*, 80(1-2):127–165, 1996.
[19] K Karagiozis, R Kamakoti, F Cirak, and C Pantano. A computational study of supersonic disk-gap-band parachutes using large-eddy simulation coupled to a structural membrane. *Journal of Fluids and Structures*, 27 (2):175–192, 2011.

[20] Yongsam Kim and Charles S Peskin. 2-D parachute simulation by the immersed boundary method. *SIAM Journal on Scientific Computing*, 28(6):2294–2312, 2006.

[21] V Lakshminarayan, C Farhat, and A Main. An embedded boundary framework for compressible turbulent flow and fluid–structure computations on structured and unstructured grids. *International Journal for Numerical Methods in Fluids*, 76(6):366–395, 2014.

[22] J Larsson, SK Lele, and P Moin. Effect of numerical dissipation on the predicted spectra for compressible turbulence. *Annual Research Briefs*, pages 47–57, 2007.

[23] John K Lin, Lauren S Shook, Joanne S Ware, and Joseph V Welch. Flexible material systems testing. 2010.

[24] Bowen Ling, Alexandre M Tartakovsky, and Ilenia Battiato. Dispersion controlled by permeable surfaces: surface properties and scaling. *Journal of Fluid Mechanics*, 801:13–42, 2016.

[25] Alex Main, Xianyi Zeng, Philip Avery, and Charbel Farhat. An enhanced FIVER method for multi-material flow problems with second-order convergence rate. *Journal of Computational Physics*, 329:141–172, 2017.

[26] Simon Mendez and Franck Nicoud. Adiabatic homogeneous model for flow around a multiperforated plate. *AIAA journal*, 46(10):2623–2633, 2008.

[27] Simon Mendez and Franck Nicoud. Large-eddy simulation of a bi-periodic turbulent flow with effusion. *Journal of Fluid Mechanics*, 598:27–65, 2008.

[28] Simon Mendez, Franck Nicoud, and Thierry Poinsot. Large-eddy simulation of a turbulent flow around a multi-perforated plate. In *Complex Effects in Large Eddy Simulations*, pages 289–303. Springer, 2007.

[29] Morris Muskat. The flow of compressible fluids through porous media and some problems in heat conduction. *Physics*, 5(3):71–94, 1934.

[30] V Nassehi. Modelling of combined Navier–Stokes and Darcy flows in crossflow membrane filtration. *Chemical Engineering Science*, 53(6):1253–1265, 1998.

[31] Taku Nonomura and Kozo Fujii. Robust explicit formulation of weighted compact nonlinear scheme. *Computers & Fluids*, 85:8–18, 2013.

[32] Taku Nonomura, Seiichiro Morizawa, Hiroshi Terashima, Shigeru Obayashi, and Kozo Fujii. Numerical (error) issues on compressible multicomponent flows using a high-order differencing scheme: Weighted compact nonlinear scheme. *Journal of Computational Physics*, 231(8):3181–3210, 2012.

[33] Francesco Panerai, Arnaud Borner, Joseph C Ferguson, Nagi N Mansour, Eric C Stern, Harold S Barnard, Alastair A MacDowell, and Dilworth Y Parkinson. X-ray micro-tomography applied to NASA’s materials research: Heat shields, parachutes and asteroids.

[34] Edward G Patton, Roger H Shaw, Murray J Judd, and Michael R Raupach. Large-eddy simulation of windbreak flow. *Boundary-Layer Meteorology*, 87(2):275–307, 1998.

[35] Jason Rabinovitch, Daniel Z Huang, Raunak Borker, Philip Avery, Charbel Farhat, Armen Derkevorkian, and Lee Peterson. Towards a validated FSI computational framework for supersonic parachute deployments. In *AIAA Aviation 2019 Forum*, page 3275, 2019.

[36] Philip L Roe. Approximate Riemann solvers, parameter vectors, and difference schemes. *Journal of computational physics*, 43(2):357–372, 1981.

[37] BE Schmidt. Compressible flow through porous media with application to injection. *California Inst. of Technology IR-FM-2014.001, Pasadena, CA, 2014.*
[38] JE Shepherd and DR Begeal. Transient compressible flow in porous materials. *NASA STI/Recon Technical Report N*, 88, 1988.

[39] Eleuterio F Toro, Michael Spruce, and William Speares. Restoration of the contact surface in the HLL-Riemann solver. *Shock waves*, 4(1):25–34, 1994.

[40] Hao Wang, Eugene S Takle, and Jinmei Shen. Shelterbelts and windbreaks: mathematical modeling and computer simulations of turbulent flows. *Annual Review of Fluid Mechanics*, 33(1):549–586, 2001.

[41] Kevin Wang, Arthur Rallu, J-F Gerbeau, and Charbel Farhat. Algorithms for interface treatment and load computation in embedded boundary methods for fluid and fluid–structure interaction problems. *International Journal for Numerical Methods in Fluids*, 67(9):1175–1206, 2011.

[42] Frank M White. *Viscous fluid flow*. McGraw-Hill New York, third edition, 2006.

[43] John D Wilson. Numerical studies of flow through a windbreak. *Journal of Wind Engineering and Industrial Aerodynamics*, 21(2):119–154, 1985.

[44] Andrew M Wissink, Richard D Hornung, Scott R Kohn, Steve S Smith, and Noah Elliott. Large scale parallel structured amr calculations using the SAMRAI framework. In *Proceedings of the 2001 ACM/IEEE conference on Supercomputing*, pages 6–6. ACM, 2001.

[45] Man Long Wong. *High-order Shock-capturing Methods for Study of Shock-induced Turbulent Mixing with Adaptive Mesh Refinement Simulations*. PhD thesis, Stanford University, 2019.

[46] Man Long Wong and Sanjiva K Lele. Multiresolution feature detection in adaptive mesh refinement with high-order shock-and interface-capturing scheme. In *46th AIAA Fluid Dynamics Conference*, page 3810, 2016.

[47] Man Long Wong and Sanjiva K Lele. High-order localized dissipation weighted compact nonlinear scheme for shock-and interface-capturing in compressible flows. *Journal of Computational Physics*, 339:179–209, 2017.