Western University

Scholarship@Western

Mechanical and Materials Engineering
Publications

Mechanical and Materials Engineering
Department

2019

A Numerical Approach for Determining the Resistance of Fine Mesh Filters

Anthony Sherratt

Western University, asherra@uwo.ca

Christopher T. DeGroot

Western University, cdegroo5@uwo.ca

Anthony G. Straatman

uwo, agstraat@uwo.ca

Domenico Santoro

USP Technologies, dsantoro@usptechnologies.com

Follow this and additional works at: <https://ir.lib.uwo.ca/mechanicalpub>

Part of the [Materials Science and Engineering Commons](#), and the [Mechanical Engineering Commons](#)

Citation of this paper:

Sherratt, Anthony; DeGroot, Christopher T.; Straatman, Anthony G.; and Santoro, Domenico, "A Numerical Approach for Determining the Resistance of Fine Mesh Filters" (2019). *Mechanical and Materials Engineering Publications*. 8.

<https://ir.lib.uwo.ca/mechanicalpub/8>

A numerical approach for determining the resistance of fine mesh filters

Anthony Sherratt, Christopher T. DeGroot, Anthony G. Straatman, and Domenico Santoro

Abstract: Characterizing the resistance of mesh filters, in terms of pressure drop as a function of flow velocity, is an important part of modeling any filtration process. Most commonly, filters are characterized experimentally, which can be costly and time-consuming. This motivates a generalized numerical approach for characterizing the resistance of mesh filters based on the flow through a representative segment of a filter. There is uncertainty, however, in the correct specification of boundary conditions such that the numerical results for flow through the small segment match the overall behaviour of the filter. In this work, an experimentally validated numerical approach is developed by examining the velocity and turbulence intensity experienced across the filter. It has been shown that the flow resistance results are not sensitive to the turbulence intensity, but depend greatly on the imposed flow velocity. Specifying the peak velocity as the boundary condition in the filter simulations resulted in a good match with experiments, while using the bulk velocity did not reproduce the experimental results.

Key words: filtration, pressure drop, computational fluid dynamics, porous media.

Résumé : La caractérisation de la résistance des filtres à mailles, en termes de perte de charge en fonction de la vitesse d'écoulement, constitue une partie importante de la modélisation de tout processus de filtration. Le plus souvent, les filtres sont caractérisés expérimentalement, ce qui peut être coûteux et prendre beaucoup de temps. D'où la nécessité d'une approche numérique généralisée pour caractériser la résistance des filtres à mailles à partir de l'écoulement à travers un segment représentatif du filtre. Cependant, il existe une incertitude quant à la spécification correcte des conditions aux limites, de sorte que les résultats numériques de l'écoulement dans le petit segment correspondent au comportement général du filtre. Dans ce travail, une approche numérique validée expérimentalement est développée en examinant la vitesse et l'intensité de turbulence ressenties à travers le filtre. Il a été démontré que les résultats de la résistance à l'écoulement ne sont pas sensibles à l'intensité de la turbulence, mais dépendent fortement de la vitesse d'écoulement imposée. La spécification de la vitesse de pointe comme condition limite dans les simulations de filtre a permis d'obtenir une bonne correspondance avec les expériences, tandis que l'utilisation de la vitesse globale n'a pas permis de reproduire les résultats expérimentaux. [Traduit par la Rédaction]

Mots-clés : filtration, chute de pression, dynamique des fluides numérique, milieux poreux.

1. Introduction

Fluid flow through porous meshes, or filters, has many practical applications in science and engineering. Filters can vary dramatically in the material from which they are made, their pore size, and their weave pattern. Generally, the application of the filter will dictate the material required for the filter. Wire mesh filters are common for electrical safety gear (Bussière et al. 2017), catalytic converters (Kołodziej et al. 2009), high efficiency heat exchangers (Kays and London 1964), and

screens for greenhouses (Teitel 2010). Cloth filters are common for surgical fabric (Chu and Rawlinson 1994), while plastic, ceramic, or metallic materials are commonly used in wastewater treatment applications (Ho and Zydney 2000; Salehi et al. 2014; Tien et al. 2014; Meireles et al. 2015; Tien and Ramarao 2017). Sepiolite (clay) is used for ultra-filtration (Wang et al. 2001). In mesh filters, the weave type also varies for different applications. Wu et al. (2005) describe a number of the most common weave types. The two most common are

Received 1 April 2018. Accepted 26 September 2018.

A. Sherratt, C.T. DeGroot, and A.G. Straatman. Department of Mechanical and Materials Engineering, University of Western Ontario, London, ON N6A 5B9, Canada.

D. Santoro. Trojan Technologies, 3020 Gore Road, London, ON N5V 4T7, Canada.

Corresponding author: Christopher T. DeGroot (email: cdegroo5@uwo.ca)

Copyright remains with the author(s) or their institution(s). Permission for reuse (free in most cases) can be obtained from [RightsLink](https://www.copyright.com/).

(i) plain weave, where the warp and weft (nonwarped) wires passed alternately over and under one another, and (ii) twill weave, where each warp wire passes alternately over two then under two weft wires, and each weft wire passes alternately over two and under two warp wires. Independent of the application, material, and weave type, all filters have the common goal of trapping and removing unwanted particles from the working fluid.

A common method for mathematically determining the net effect of flow passing through a porous medium is the use of Darcy's law, which states that the pressure drop is linearly proportional to the velocity for low Reynolds numbers ($Re_d < 1$, where d is the characteristic pore dimension). For higher Reynolds number flows, the pressure drop becomes quadratic in terms of velocity and the flow is considered to be in the Forchheimer flow regime (Vafai and Tien 1981; Skjetne and Auriault 2014; Nield and Bejan 2017). The most common method for characterizing the flow resistance for different types of mesh filters is to run experiments. Experiments in the literature describe the general approach of a pump or compressor driving the working fluid (most commonly air or water) through a horizontal pipe or square tube, allowing the working fluid to flow through the mesh filter being characterized (Wu et al. 2005; Kołodziej et al. 2009; Sun et al. 2015; Bussire et al. 2017; Zhu et al. 2017). Pressure transducers are placed before and after the mesh filter to measure the pressure differential, while a flow meter is used to determine the volumetric flow rate through the experimental apparatus. The data are then used to calculate the resistance coefficients. The experimental technique can, however, be costly and time-consuming.

Another approach for investigating flow phenomena in porous filters is the use of computational fluid dynamics (CFD), which can be faster and less expensive than experiments. Sun et al. (2015) proposed a method for characterizing the pressure across wire filters. They described three models, (i) a three-dimensional model that resolved the true geometry of the wire weaves, (ii) a simplified three-dimensional model with interwoven orthogonal wires, and (iii) a simplified two-dimensional model using only horizontally placed wires. Using these three models, Sun et al. (2015) found good agreement between experimental data and numerical predictions. They also found that the simplification of the mesh geometry had little effect on the pressure drop across the varying meshes while greatly reducing computational time. However, an issue arises when attempting to apply the two dimensional model to other mesh types. Given that there is inherently an infinite pore size when there are no vertical threads, there is no clear method for incorporating the thread spacing for a given mesh. This means that an experiment must be run for each mesh that needs to be characterized. Also,

the boundary conditions implemented in this study were not clear, motivating further study of this aspect.

Teitel (2010) proposed a different method of reducing the complexity and computational time for CFD simulations of woven screens. It was hypothesized that by modeling mesh screens as a porous jump, the computational time would be reduced substantially compared to a three-dimensional model of the screen, with limited effect on the resulting pressure drop. Two models were compared, (i) a three-dimensional model of the mesh screen and (ii) a porous jump where the screen permeability and inertial factor were calculated using various empirical correlations. It was found that the porous jump did significantly reduce the computational time, as the porous jump can reach grid independence with a very coarse mesh. However, depending on the method for calculating the permeability and inertial factor, the results ranged from very good to very poor compared to the three-dimensional model (which matched experimental results very well) and experimental data. This range of results depended on the modeler's knowledge of the screen, such as the correct thickness ratio between the filter and the porous jump modeled. Therefore, depending on the application and screen type, experiments may still be needed to determine the correct empirical correlations for the permeability and inertial factors used in the porous jump model.

The present study proposes a generalized method for efficiently determining the pressure drop across mesh filters using CFD simulations on a representative segment of an idealized filter geometry. First, the flow in the inlet section that leads up to the filter is computed and examined in terms of the dimensionless velocity and turbulence intensity profiles. These data are then used to develop boundary conditions for the representative filter segment. It will be shown, using experimental data for validation, that the peak velocity should be used as the boundary condition in the filter simulations.

2. Theory

The pressure drop across a porous medium is described by Darcy's law, given as

$$(1) \quad \frac{\Delta P}{L} = \frac{\mu U}{K}$$

where ΔP is the pressure differential, L is the thickness of the medium, μ is the dynamic viscosity, U is the bulk fluid velocity, and K is the permeability (Skjetne and Auriault 2014; Nield and Bejan 2017; Zhu et al. 2017). Equation (1) may also be written in terms of the flow resistance, R , which includes the lumped effects of the medium thickness, as

$$(2) \quad \Delta P = \mu UR$$

The permeability, K , and resistance, R , are only constant for low Reynolds numbers based on pore

diameter ($Re_d < 1$). When considering flows at higher Reynolds numbers, the flow through porous media transitions into the Forchheimer flow regime (Vafai and Tien 1981; Teitel 2010; Skjetne and Auriault 2014; Nield and Bejan 2017). Rather than the pressure drop being linearly proportional to the velocity, there is also a quadratic term, and the expression becomes

$$(3) \quad \frac{\Delta P}{L} = \frac{\mu U}{K} + C_f \rho \frac{U^2}{\sqrt{K}}$$

where ρ is the fluid density and C_f is a constant that depends on the pore geometry. The resistance term, R , in eq. (2) can also be modified to show the same quadratic dependence, by writing it as

$$(4) \quad R = a + bU$$

where a and b are referred to as resistance coefficients.

3. Experimental methods

A schematic diagram of the experimental setup is given in Fig. 1. A centrifugal pump delivers water to the inlet pipe through a soft polyvinyl chloride (PVC) tube connected at the inlet. The water then flows through the rigid PVC inlet pipe of 2.5 m length towards the filter adapter, where two square pieces of PVC are bolted together to form a sealed flange. This secures the mesh filter being tested. Based on the pipe diameter, 0.041 m, and flow rate, 0.3 to 1.5 m/s, the Reynolds number based on diameter ranges from 12000 to 61000, making the flow fully turbulent ($Re_d > 2300$) (Moran et al. 2014). The length of the pipe was selected to ensure a fully developed flow at the filter location. The filter adapter was connected to the outlet pipe, again rigid PVC, with the same diameter as the inlet pipe. The outlet was attached to another portion of soft PVC tube that directed the water into an 80 L reservoir. The flow rate was measured by timing water flowing into a fixed volume, then weighing the water afterwards.

A Gould NPE 1HP pump was used to provide a consistent flow over a wide range of flow rates. The Gould NPE pump is a centrifugal pump controlled by an SMVector variable frequency drive (VFD). An OMEGA digital pressure gauge was used to measure the pressure drop across the mesh filter. The pipe diameter was constant across the mesh, such that the total pressure drop was equal to the static pressure difference across the mesh. The digital pressure gauge measures the pressure in millibar (mbar), with an accuracy of 0.5 mbar. The static pressure of 1 m of water was tested before each experiment, to ensure proper calibration of the sensor. Table 1 gives a summary of the relevant specifications for the experimental apparatus.

Each mesh filter was characterized in three complete experiments to observe the variations between separate tests. Each filter was characterized for the experiment

Fig. 1. Schematic diagram of experimental setup.

Table 1. Experiment specifications.

Component	Specification
Inlet pipe	Length = 2.44 m, Diameter = 0.041 m
Outlet pipe	Length = 1.52 m, Diameter = 0.041 m
Water reservoir	80 L
Velocity range	0.3–1.5 (m/s)
Pump	Gould 1HP (745.7 W) centrifugal pump
VFD controller	SMVector, 1.5HP (1118.55 W)

Table 2. Geometric properties of tested mesh filters.

Nominal pore size (μm)	Thread diameter (μm)	Open area (%)
158	145	26
350	250	34

based upon two parameters, thread diameter and open area percentage. The properties of the filters for each nominal pore size are given in Table 2.

4. Numerical simulations

To determine the inflow characteristics for flow through the mesh filter, a numerical study was first performed on the pipe flow to determine the velocity profile and turbulence quantities experienced at the filter. Using these data, boundary conditions for a representative element of the filter were determined for a second numerical study, from which results are compared to the experimental data.

4.1. Mesh filter geometric model

Two different mesh filters were used in this experiment with nominal pore sizes of 158 μm and 350 μm, as shown in Fig. 2. Due to the complexity of the mesh filter being modeled, an assumption was made about the size of the domain in relation to the filter pore. When considering the size of the pores (350 μm for example) and

Fig. 2. Microscope images of meshes with nominal pore sizes of (a) 158 μm and (b) 350 μm . [Colour online.]

assuming the pore geometry to be square, there would be more than 2000 pores if the domain were the entire mesh filter insert. With this number of pores, and each pore needing on average 150,000 control volumes to achieve grid independence, the simulation time would be excessive. Therefore, it is more sensible to consider a representative part of the filter, assuming that the number of pores is large enough that the effects of the pipe wall are minimal.

Therefore, the mesh filter was modeled using a single, spatially periodic pore. Symmetry boundary conditions were used to replicate the effects of the surrounding pores in the mesh filter insert. This simplification reduced computational time while still providing an accurate representation of the pressure drop that would be seen across the entire filter. When observing the filter meshes under a microscope, as in Fig. 2, it is clear that while there is a repeating pattern, there are slight geometric differences from pore to pore. However, the mesh filter geometry was idealized as square, with average dimensions obtained by taking the known thread diameter and open area percentage provided by the manufacturer. The width of the domain can be calculated, corresponding to the average pore geometry within the actual filter. It was also assumed in the idealized geometric model that the threads are circular and that thread junctions are as shown in Fig. 3a. Figure 2 shows that the weave type, on average, is plain weave, and there are locations along the mesh filter where it appears to be twill weave. It is also shown that the mesh thread deforms somewhat when the mesh is being manufactured. Since these geometric nuances would be difficult to incorporate, and it is hypothesized that their effect on the pressure drop is negligible, it was assumed that they all average out to the idealized geometry shown in Fig. 3a. This is a common approach found throughout the literature (Teitel 2010; Sun et al. 2015). The simulation domain is shown in Fig. 3b, where the inflow section has a length of $2D$ and the outflow section has a length of $5D$, where D is the thread diameter.

Fig. 3. Diagrams of (a) idealized mesh filter model with no geometric weave and (b) cross-section of computational domain with relevant dimensions in terms of thread diameter, D . [Colour online.]

4.2. Governing equations

4.2.1. Conservation equations

The flows considered in the present study involve incompressible Newtonian fluids, where the Reynolds Averaged Navier–Stokes equations governing the time-averaged velocity and pressure fields are given as (Schlichting and Gersten 2017)

$$(5) \quad \frac{\partial}{\partial x_i} (\bar{u}_i) = 0$$

and

$$(6) \quad \frac{\partial(\rho \bar{u}_i)}{\partial t} + \frac{\partial(\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = -\frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij}) - \frac{\partial}{\partial x_j} (\rho \bar{u}_i' \bar{u}_j')$$

where $\mathbf{x}_i$ is the position vector equal to $[x, y, z]$; $\bar{\mathbf{u}}_i$ and $\bar{P}$ denote the time-averaged velocity vector and time-averaged pressure, respectively; $\mathbf{u}'_i$ is the turbulent velocity fluctuation vector and $\rho \mathbf{u}'_i \mathbf{u}'_j$ denotes the Reynolds stress tensor. The laminar stress tensor, $\bar{\tau}_{ij}$, is given as

$$(7) \quad \bar{\tau}_{ij} = \mu \left[\frac{\partial \bar{\mathbf{u}}_i}{\partial \mathbf{x}_j} + \frac{\partial \bar{\mathbf{u}}_j}{\partial \mathbf{x}_i} \right]$$

To close eq. (6), a turbulence model is required to calculate the Reynolds stress term.

4.2.2. Turbulence modeling

The standard $k-\epsilon$ turbulence model, with enhanced wall functions, was used to model turbulence in the present study. The enhanced wall function model ensures that the model is robust to mesh refinement, since it allows for the near-wall region to be resolved when the mesh is fine enough to compute the viscous sublayer near the walls. The transport equations for the incompressible $k-\epsilon$ turbulence model, in the absence of buoyancy and user source terms, are given as (Lauder and Sharma 1974)

$$(8) \quad \frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial \mathbf{x}_i}(\rho k \bar{\mathbf{u}}_i) = \frac{\partial}{\partial \mathbf{x}_j} \left[\left(\mu + \left(\frac{\mu_t}{\sigma_k} \right) \right) \left(\frac{\partial k}{\partial \mathbf{x}_j} \right) \right] + G_k - \rho \epsilon$$

$$(9) \quad \frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial \mathbf{x}_i}(\rho \epsilon \bar{\mathbf{u}}_i) = \frac{\partial}{\partial \mathbf{x}_j} \left[\left(\mu + \left(\frac{\mu_t}{\sigma_\epsilon} \right) \right) \left(\frac{\partial \epsilon}{\partial \mathbf{x}_j} \right) \right] + C_{1\epsilon} \left(\frac{\epsilon}{k} \right) (G_k + C_{3\epsilon} G_b) - C_{2\epsilon} \rho \left(\frac{\epsilon^2}{k} \right)$$

where $C_{1\epsilon}$, $C_{2\epsilon}$, C_μ , σ_k , and σ_ϵ are turbulence model constants and ϵ is the Eddy dissipation rate.

The Reynolds stress term is modelled using the Boussinesq approximation, given as

$$(10) \quad (\rho \mathbf{u}'_i \mathbf{u}'_j) = \mu_t \left[\frac{\partial \bar{\mathbf{u}}_i}{\partial \mathbf{x}_j} + \frac{\partial \bar{\mathbf{u}}_j}{\partial \mathbf{x}_i} \right] - \frac{2}{3} \rho k \delta_{ij}$$

where μ_t is the eddy viscosity, k is the turbulent kinetic energy, and δ_{ij} is the Kronecker delta. The turbulent eddy viscosity is computed as

$$(11) \quad \mu_t = \rho C_\mu \left(\frac{k^2}{\epsilon} \right)$$

The definition of the turbulence production, G_k , and the values of the remaining turbulence constants in eqs. (8) and (9) are the default values in ANSYS® Fluent, which is used to solve the sets of equations listed previously. These values and definitions can be found in the software user manual (ANSYS Inc. 2015).

5. Results

5.1. Inlet pipe simulations

In the idealized mesh filter model, only a single pore is modeled, under the implicit assumption that the pipe

cross-section is large enough that there are a large number of pores contained within the filter insert. To conduct simulations of the representative filter element, boundary conditions for velocity and turbulence intensity must be specified at the inlet. However, there is a distribution of both turbulent kinetic energy and velocity across the filter insert, and these distributions are not generally known. To determine the correct turbulence and velocity boundary conditions for the mesh filter model, further examination of the inlet pipe flow was required.

The inlet pipe model was generated in SOLIDWORKS® 2017 and then imported into ANSYS Workbench. The dimensions of the model are the same as in the experiment, with a length of 2.44 m and diameter of 0.041 m. Because the flow is symmetric about the centre of the pipe, only one quarter of the pipe was modelled, with symmetry conditions applied on the cut planes. On the pipe walls, no-slip boundary conditions were applied. At the inlet, a uniform velocity equal to the bulk velocity from the experiments was specified. The turbulent kinetic energy and eddy dissipation rate were calculated, based on a 5% turbulence intensity and the given pipe diameter. The outlet was specified to zero gauge pressure. The discretized governing equations were solved using the commercial CFD software ANSYS® Fluent, Release 17.2. Meshing was done using ANSYS Meshing and a grid independence study was done to ensure results did not change by more than 1% between subsequent grids, as shown in Table 3. The fine grid was used for all subsequent calculations. The discretization methods chosen for the simulations can be found in Table 4.

To produce a general set of boundary conditions for the mesh filter model from the inlet pipe model, the results for the turbulent kinetic energy and velocity profiles needed to be comparable between different simulations. This was done by normalizing the turbulent kinetic energy into the turbulence intensity, which is defined as

$$(12) \quad I = \frac{1}{U} \sqrt{\frac{2k}{3}}$$

where I is the turbulence intensity and U is bulk fluid velocity specified at the inlet. The velocity profile was normalized using the pipe's bulk velocity, according to

$$(13) \quad u^* = \frac{u}{U}$$

where u^* is the dimensionless velocity at a given location and u is the measured velocity at a given location.

Figures 4 and 5 show the results for the dimensionless velocity and turbulence intensity profiles as functions of the dimensionless radial coordinate, respectively. As expected, the dimensionless velocity profile collapses onto a single curve (Schlichting and Gersten 2017).

Table 3. Results of grid independence test on inlet pipe for velocity and turbulent kinetic energy profiles, where coarse and fine correspond to 84375 and 159375 elements within the domain.

Radial position (m)	Velocity (m/s)			Turbulent kinetic energy (m^2/s^2)		
	Coarse	Fine	% Difference	Coarse	Fine	% Difference
0.00108	0.59613	0.59610	0.005	0.00108	0.00109	0.9
0.00538	0.58891	0.58890	0.001	0.00123	0.00124	0.8
0.00968	0.57349	0.57352	0.005	0.00151	0.00152	0.7
0.01399	0.53363	0.53365	0.003	0.00205	0.00205	<0.5
0.01829	0.45509	0.45509	<0.002	0.00290	0.00290	<0.3

Table 4. Parameters and discretization schemes chosen for CFD models.

Category	Setting	Value
General	Solver	Pressure-based, steady, absolute reference frame
Models	Energy	Isothermal
	Viscous	Turbulent ($k-\epsilon$ with enhanced wall function)
Materials Solution methods	Fluids	Water
	Pressure-velocity coupling scheme	SIMPLE
	Gradient	Least squares cell-based
	Pressure	Second order
Residuals	Momentum and turbulence advection	Second order upwind
	Scaling	Local scaling
	Convergence criteria	1.0×10^{-5}

Therefore, the velocity experienced at a particular location on the mesh filter insert can be determined based on these results. On the other hand, the results for turbulence intensity do not collapse onto a single curve. However, the turbulence intensity is generally bounded between 5% and 10% for all radial locations within the pipe. Therefore, it is anticipated that a single uniform value of turbulence intensity may be suitable, provided the pressure loss is not strongly affected by the inflow turbulence intensity.

5.2. Simulations of representative element of mesh filter

5.2.1. Grid independence

After the pipe flow that is experienced by the mesh filter was characterized, simulations on the representative filter element, shown in Fig. 3, were conducted. Standard no-slip boundary conditions were applied on the thread surfaces. At the inlet, specified values of velocity and turbulence intensity were applied. At the outlet, zero gauge pressure was specified. Symmetry boundary conditions were applied on all of the remaining outer surfaces of

Fig. 4. Normalized velocity profile at pipe outlet for varying inlet velocities.

Fig. 5. Turbulence intensity profile at pipe outlet for varying inlet velocities.

the domain. Discretization methods were the same as those for the inlet pipe simulations, which are summarized in Table 4. For both the 158 μm and 350 μm mesh filter geometries, grid independence tests were conducted. The grid was refined to 1.5%, as seen in Tables 5 and 6, for the 158 μm and 350 μm mesh filters,

Table 5. Grid independence test for 158 μm mesh.

	No. of control volumes	Pressure drop (Pa)	Change (%)
Coarse	50002	7209.55	N/A
Medium	75485	7356.66	2.0
Fine	160185	7386.10	0.4

Table 6. Grid independence test for 350 μm mesh.

	No. of control volumes	Pressure drop (Pa)	Change (%)
Coarse	25251	1693.80	N/A
Medium	45178	1824.38	7.7
Fine	61428	1802.86	1.2

respectively. The fine mesh was used for both the 158 μm and 350 μm mesh filters for all subsequent calculations.

5.2.2. Sensitivity analysis

In the CFD model of the representative element of the mesh filter, the turbulence intensity is given as a boundary condition at the inlet. To test the effect of turbulence intensity on pressure drop, the turbulence intensity was varied from 1% to 10% on the 350 μm mesh filter model, with 5% being the reference value used for the grid independence tests described previously. A constant inlet velocity of 0.5 m/s was used for this sensitivity study. The results in Table 7 show that the inlet turbulence intensity has little effect on the predicted pressure drop across the mesh filter, over the range of intensities considered. This result is expected due to the high turbulence production within the filter region. Therefore, the inlet turbulence intensity was held constant at 5% for all subsequent calculations.

It can be seen in Fig. 4 that the velocity experienced by a mesh filter contained within a pipe is generally bounded between $0.9U$ and $1.2U$ for the majority of the pipe radius, apart from the narrow region very near the pipe wall. This transition range from $0.9U$ to 0 covers a small area and therefore contributes little to the pressure drop across the mesh filter. Therefore, a sensitivity analysis on the inlet velocity imposed on the representative filter element was conducted, and the results were compared with the pressure drop measured from experiments. The best match between experimental and numerical results was obtained when the boundary condition imposed on the filter element was the peak velocity within the corresponding pipe velocity profile. When the bulk velocity was applied as the boundary condition, significant errors between the numerical and experimental results were observed. Physically, it can be reasoned that the majority of the filter experiences a velocity that is closer to the peak velocity than the bulk

Table 7. Sensitivity analysis of effects of inflow turbulence intensity on pressure drop across mesh filter, using 5% turbulence intensity as reference value.

Turbulence intensity (%)	$P_{\text{inlet}} - P_{\text{outlet}}$	Difference (%)
1	5302.52	0.26
2	5304.6	0.22
3	5307.4	0.17
4	5310.8	0.10
5	5316.23	—
6	5320.87	0.09
7	5325.9	0.18
8	5331.25	0.28
9	5336.91	0.39
10	5340.47	0.46

velocity, according to the results shown in Fig. 4. Therefore, the peak velocity was used as the velocity boundary condition for all subsequent calculations on the representative filter elements.

5.2.3. Comparison of numerical and experimental results

Knowing that the inflow turbulence has minimal effect on the pressure drop results and that the experimental results can be best matched by imposing the peak pipe flow velocity as the boundary condition for the simulations of the representative filter element, further simulations were run for both filter pore sizes across a range of flow velocities. Experiments were also conducted for similar ranges of flow velocities for both filter sizes. All flow experiments were repeated in triplicate to show that the results were consistent and repeatable. Results for both experimental and numerical pressure drops, as functions of velocity, are shown in Fig. 6. It should be noted that because the velocity imposed in the pore-level simulations was the peak pipe velocity, $1.2U$, numerical results in Fig. 6 have been converted to the equivalent bulk velocity, U . It is clear from the figure that the results from the CFD analysis compare very well to the experimental values, because all CFD predictions are within the experimental deviations. It can also be seen that the 158 μm mesh filter simulations compare almost exactly to the mean experimental values, while the 350 μm mesh filter results compare well with the low end of the experimental deviation. This small deviation may be due to the variability in the pore sizes and an imperfect characterization of the filter geometry and dimensions. If the actual filter size were to be smaller than what was estimated, it would make sense for the experimental results to show higher pressure drop. These results are quite acceptable, because they are within the variability of the experimental data. Therefore, it is concluded that the proposed CFD-based approach can be used to accurately compute the

Fig. 6. Experimental results for pressure drop as a function of bulk flow velocity in a pipe flow with filter insert, compared to numerical results for pressure drop in a representative filter element for mesh filters with nominal pore sizes of 158 μm and 350 μm . [Colour online.]

Table 8. Resistance coefficients for 158 μm and 350 μm mesh, from CFD results.

Nominal pore size (μm)	a	b
158	1032000	9306000
350	30900	3634000

pressure drop across mesh filters using a highly idealized weave pattern.

As expected, the pressure drop follows a quadratic trend with respect to fluid velocity, confirming that the flow is within the Forchheimer flow regime. Referring to eq. (2) and substituting eq. (4) for the resistance term, the resistance coefficients a and b can be determined from the CFD simulation results. The a and b values for the 158 μm and 350 μm mesh filters can be found in Table 8. These values can then be used to easily calculate the pressure drop across a mesh filter for a known flow velocity. Similarly, these coefficients could be implemented within the context of a porous jump surface, as implemented by Teitel (2010), which would allow for efficient simulation of flow through filters in the context of larger CFD simulations.

6. Summary

In this study, an experimentally validated numerical model was developed to characterize the resistance across mesh filters using CFD simulations of a small, idealized element of the filter. Boundary conditions for the filter element were developed by considering the flow and turbulence quantities in a fully-developed pipe flow. A sensitivity analysis revealed that the pressure drop is not significantly affected by the imposed turbulence intensity, but that the imposed velocity has a large effect. Results have shown that specifying the maximum

velocity experienced by the filter as the inlet boundary condition for the filter element provides the best match with experimental data, while imposing the bulk velocity results in significant error.

Acknowledgements

This work was supported by the Natural Science and Engineering Research Council (NSERC) (CRDPJ 488704-15, 2017) and (EGP 496723-16, 2016).

References

- ANSYS Inc. 2015. Fluent theory guide, version 17.2. ANSYS Inc., Canonsburg, PA.
- Bussière, W., Rochette, D., Clain, S., André, P., and Renard, J.B. 2017. Pressure drop measurements for woven metal mesh screens used in electrical safety switchgears. *Int. J. Heat Fluid Flow*, **65**: 60–72. doi:10.1016/j.ijheatfluidflow.2017.02.008.
- Chu, C.C., and Rawlinson, J. 1994. Mathematical modeling of water permeability of surgical fabrics for vascular use. *J. Biomed. Mater. Res.* **28**(4): 441–448. doi:10.1002/jbm.820280406. PMID:8006049.
- Ho, C.C., and Zydney, A.L. 2000. A combined pore blockage and cake filtration model for protein fouling during microfiltration. *J. Colloid Interface Sci.* **232**(2): 389–399. doi:10.1006/jcis.2000.7231. PMID:11097775.
- Kays, W.M., and London, A.L. 1964. Compact heat exchangers. 2nd ed. McGraw-Hill, New York, NY.
- Kołodziej, A., Jaroszyński, M., Janus, B., Kleszcz, T., Łojewska, J., and Łojewski, T. 2009. An experimental study of the pressure drop in fluid flows through wire gauzes. *Chem. Eng. Commun.* **196**(8): 932–949. doi:10.1080/00986440902743851.
- Launder, B.E., and Sharma, B.I. 1974. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. *Lett. Heat Mass Transfer*, **1**(2): 131–137. doi:10.1016/0094-4548(74)90150-7.
- Meireles, M., Prat, M., and Estachy, G. 2015. Analytical modeling of steady-state filtration process in an automatic self-cleaning filter. *Chem. Eng. Res. Design*, **100**: 15–26. doi:10.1016/j.cherd.2015.04.030.
- Moran, M.J., Shapiro, H.N.D., Boettner, D., and Bailey, M.B. 2014. Fundamentals of engineering thermodynamics. 8th ed. Wiley, Hoboken, NJ, USA.
- Nield, D.A., and Bejan, A. 2017. Mechanics of fluid flow through a porous medium. Convection in porous media. Springer, New York, NY, USA. pp. 1–35.
- Salehi, E., Madaeni, S.S., Shamsabadi, A.A., and Laki, S. 2014. Applicability of ceramic membrane filters in pretreatment of coke-contaminated petrochemical wastewater: economic feasibility study. *Ceram. Int.* **40**(3): 4805–4810. doi:10.1016/j.ceramint.2013.09.029.
- Schlichting, H., and Gersten, K. 2017. Boundary layer theory. Springer-Verlag, Berlin, Germany.
- Skjetne, E., and Auriault, J.-L. 2014. High-velocity laminar and turbulent flow in porous media. *CEUR Workshop Proc.* **1225**: 41–42.
- Sun, H., Bu, S., and Luan, Y. 2015. A high-precision method for calculating the pressure drop across wire mesh filters. *Chem. Eng. Sci.* **127**: 143–150. doi:10.1016/j.ces.2015.01.023.
- Teitel, M. 2010. Using computational fluid dynamics simulations to determine pressure drops on woven screens. *Biosyst. Eng.* **105**(2): 172–179. doi:10.1016/j.biosystemseng.2009.10.005.
- Tien, C., and Ramarao, B.V. 2017. Modeling the performance of cross-flow filtration based on particle adhesion. *Chem. Eng. Res. Des.* **117**: 336–345. doi:10.1016/j.cherd.2016.09.011.

- Tien, C., Ramarao, B.V., and Yasarla, R. 2014. A blocking model of membrane filtration. *Chem. Eng. Sci.* **111**: 421–431. doi:10.1016/j.ces.2014.01.022.
- Vafai, K., and Tien, C.L. 1981. Boundary and inertia effects on flow and heat transfer in porous media. *Int. J. Heat Mass Transfer*, **24**(2): 195–203. doi:10.1016/0017-9310(81)90027-2.
- Wang, Q.K., Matsuura, T., Feng, C.Y., Weir, M.R., Detellier, C., Rutadinka, E., and Van Mao, R.L. 2001. The sepiolite membrane for ultrafiltration. *J. Membr. Sci.* **184**(2): 153–163. doi:10.1016/S0376-7388(00)00605-0.
- Wu, W.T., Liu, J.F., Li, W.J., and Hsieh, W.H. 2005. Measurement and correlation of hydraulic resistance of flow through woven metal screens. *Int. J. Heat Mass Transfer*, **48**(14): 3008–3017. doi:10.1016/j.ijheatmasstransfer.2005.01.038.
- Zhu, Z., Wang, Q., and Wu, Q. 2017. On the examination of the Darcy permeability of soft fibrous porous media; new correlations. *Chem. Eng. Sci.* **173**: 525–536. doi:10.1016/j.ces.2017.08.021.

List of Symbols

a	resistance coefficient (1/m)	G_k	turbulence production
b	resistance coefficient (s/m ²)	I	turbulence intensity
C_f	Forchheimer flow constant	K	Darcy permeability (m ²)
C_{1e}	turbulence model constant	k	turbulent kinetic energy (m ² /s ²)
C_{2e}	turbulence model constant	L	thickness of medium (m)
C_μ	turbulence model constant	$\bar{P}$	time-averaged pressure (Pa)
d	characteristic pore diameter (m)	ΔP	pressure differential (Pa)
		R	flow resistance (1/m)
		Re_d	Reynolds number based on diameter
		ϵ	Eddy dissipation rate (m ² /s ³)
		U	bulk fluid velocity (m/s)
		u	measured velocity at a given location (m/s)
		u^*	dimensionless velocity
		u_i'	turbulent velocity fluctuation vector (m/s)
		$\bar{u}_i$	time-averaged velocity vector, equal to $[u,v,w]$ (m/s)
		x_i	position vector, equal to $[x,y,z]$ (m)
		δ_{ij}	Kronecker delta
		μ	dynamic viscosity of fluid (Pa·s)
		μ_t	turbulent (Eddy) viscosity (Pa·s)
		ρ	fluid density (kg/m ³)
		σ_k	turbulence model constant
		σ_ϵ	turbulence model constant
		$\bar{\tau}_{ij}$	time-averaged laminar stress tensor (N/m ²)