A Numerical Approach for Determining the Resistance of Fine Mesh Filters

Anthony Sherratt\textsuperscript{a}, Christopher T. DeGroot\textsuperscript{a,*}, Anthony G. Straatman\textsuperscript{a}, Domenico Santoro\textsuperscript{b}

\textsuperscript{a}Department of Mechanical and Materials Engineering, University of Western Ontario, London, Ontario, Canada
\textsuperscript{b}Trojan Technologies, 3020 Gore Road, London, Ontario, Canada

\textsuperscript{*}Corresponding author

Email address: cdegroo5@uwo.ca (Christopher T. DeGroot)
Abstract

Characterizing the resistance of mesh filters, in terms of the 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 the need for a generalized numerical approach for characterizing the resistance of mesh filters based on the flow through a representative segment of the 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 was not able to reproduce the experimental results.

Keywords: Filtration, Pressure Drop, Computational Fluid Dynamics, Porous Media
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 (Meireles et al. 2015; Tien et al. 2014; Tien and Ramarao 2017; Ho and Zydney 2000; Salehi et al. 2014). 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, with the two most common being: (i) plain weave, where the warp and weft (non-warped) wires passed alternately over and under one another and (ii) twill weave, where each warp wire passes alternatively over two then under two weft wires, and each weft wire passes alternately over two and under two warp wires. Depending on 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 through 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; Nield and Bejan 2017; Skjetne and Auriault 2014). The most common method for characterizing the flow resistance for different types of mesh filters is to run experiments. Experiments in 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 (Bussière et al. 2017; Kołodziej et al. 2009; Wu et al. 2005; Zhu et al. 2017; Sun et al. 2015). 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 is 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 then means an experiment needs to 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 when compared to a 3-D model of the screen, with limited effect on the resulting pressure drop. Two models were compared: (i) a 3-D 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, when compared to the 3-D (which matched experimental results very well) and experimental data, ranged from very good to very poor. This range of results depended on the knowledge the modeler had of the screen, for example, knowing the correct thickness ratio between the
filter and the porous jump modeled. Therefore, depending on the application and screen type, experiments may still need to be done 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. This data is 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

\[ \frac{\Delta P}{L} = \frac{\mu U}{K} \]

where \( \Delta P \) is the pressure differential, \( L \) is the thickness of the medium, \( \mu \) is the dynamic viscosity, \( U \) is the bulk fluid velocity, and \( K \) is the permeability (Nield and Bejan 2017; Skjetne and Auriault 2014; 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

\[ \Delta P = \mu U R \] (2)

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 (Teitel 2010; Vafai and Tien 1981; Nield and Bejan 2017; Skjetne and Auriault 2014). Rather than the pressure drop being linearly proportional to the velocity, there is also a quadratic term, and the expression becomes

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

\[
R = a + bU
\]  
(4)

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 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 was used to secure 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 of 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 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

In order 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 this 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 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 with regards to the filter pore. When considering the size of the pores (350 µm for example) and assuming the pore geometry to be square, there would be more than 2000 pores if the domain was to be 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 shown 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.
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. 3(a). Figure 2 shows that the weave
type, on average, is plain weave and there are locations along the mesh filter were 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. 3(a). This is a common approach
found throughout literature (Teitel 2010; Sun et al. 2015). The simulation domain is shown
in Fig. 3(b), where the inflow section has a length of 2D and the outflow section has a length
of 5D, where D is the thread diameter.

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 (RANS) equations governing the time-averaged veloc-
ity and pressure fields are given as (Schlichting and Gersten 2017)

\[ \frac{\partial}{\partial x_i} (\bar{u}_i) = 0 \]  \hspace{1cm} (5)

and

\[ \frac{\partial (\rho \bar{u}_i)}{\partial t} + \frac{\partial (\rho \bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{\partial \bar{P}}{\partial x_i} + \frac{\partial}{\partial x_j} (\tau_{ij}) - \frac{\partial}{\partial x_j} (\rho \bar{u}_i' \bar{u}_j') \] \hspace{1cm} (6)

where \( \bar{u}_i \) and \( \bar{P} \) denote the time-averaged velocity vector and time-averaged pressure, respec-
tively, \( u'_i \) is the turbulent velocity fluctuation vector and \( \rho u'_i' u'_j' \) denotes the Reynolds stress
tensor. The laminar stress tensor, \( \tau_{ij} \), is given as

\[ \tau_{ij} = \mu \left[ \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right] \] \hspace{1cm} (7)

In order 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 (Launder and Sharma 1974)

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_k} \right) \left( \frac{\partial k}{\partial x_j} \right) \right] + G_k - \rho \epsilon \tag{8}$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_i}(\rho \epsilon u_i) = \frac{\partial}{\partial x_j} \left[ \left( \frac{\mu}{\sigma_\epsilon} \right) \left( \frac{\partial \epsilon}{\partial x_j} \right) \right] + C_{1\epsilon} \left( \frac{\epsilon}{k} \right) (G_k + C_3 \epsilon G_b) - C_{2\epsilon} \rho \left( \frac{\epsilon^3}{k} \right) \tag{9}$$

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

$$\langle \rho u'_i u'_j \rangle = \mu_t \left[ \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right] - \frac{2}{3} \rho k \delta_{ij} \tag{10}$$

where $\mu_t$ is the eddy viscosity, $k$ is the turbulent kinetic energy, and $\delta_{ij}$ is the Kronecker delta. The turbulent eddy viscosity is computed as

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

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. In order 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. Since 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

\[ I = \frac{1}{U} \sqrt{\frac{2k}{3}} \]  

(12)
where $I$ is the turbulence intensity, $U$ is bulk fluid velocity specified at the inlet and $k$ is the turbulent kinetic energy. The velocity profile was normalized using the pipe’s bulk velocity, according to

$$u^* = \frac{u}{U}$$

where $u^*$ is the dimensionless velocity at a given location, $u$ is the measured velocity at a given location, and $U$ is the bulk inlet velocity.

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). 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, it is shown that 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 characterizing the pipe flow that is experienced by the mesh filter, 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 the domain. Discretization methods are the same as those for the inlet pipe simulations, which are summarized in Table 4. For both the 158 and 350 µm mesh filter geometries, a grid independence tests were conducted. The grid was refined to 1.5%, as seen in Table 5 and 6, for the 158 and 350 µm mesh filters, respectively. The fine mesh was used for both the 158 and 350 µm 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 affect 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. It was found through this study that 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 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 are 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 since the velocity imposed in the pore-level simulations was the peak pipe velocity (i.e. \(1.2U\)), 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, since all CFD predictions are within the experimental deviations. It can also be seen that the the 158 \(\mu\)m mesh filter simulations compare almost exactly to the mean experimental values, while the 350 \(\mu\)m mesh filter results compare well with the low-end of the experimental deviation. The reason for this small deviation may be due to the variability in the pore sizes, as well as 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. In any case, these results are found to be quite acceptable, since 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 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 determined from the CFD simulation results. The \(a\) and \(b\) values for the 158 and 350 \(\mu\)m 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 within 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 effected 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

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


amic membrane filters in pretreatment of coke-contaminated petrochemical wastewater:


Media. CEUR Workshop Proceedings, 1225:41–42.

Sun, H., Bu, S., and Luan, Y. 2015. A high-precision method for calculating the pressure

Teitel, M. 2010. Using computational fluid dynamics simulations to determine pressure

Tien, Chi and Ramarao, Bandaru V. 2017. Modeling the performance of cross-flow filtration


Vafai, K and Tien, C.L. 1981. Boundary and inertia effects on flow and heat transfer in

Wang, Q. K., Matsuura, T., Feng, C. Y., Weir, M. R., Detellier, C., Rutadinka, E., and

hydraulic resistance of flow through woven metal screens. International Journal of Heat

Nomenclature

\( a \), Resistance Coefficient, \([1/m]\)

\( b \), Resistance Coefficient, \([s/m^2]\)

\( C_f \), Forchheimer Flow Constant

\( C_{1\epsilon} \), Turbulence Model Constant

\( C_{2\epsilon} \), Turbulence Model Constant

\( C_\mu \), Turbulence Model Constant

\( D \), Diameter, \([m]\)

\( G_k \), Turbulence Production

\( k \), Turbulent Kinetic Energy, \( [m^2/s^2] \)

\( K \), Darcy Permeability, \([m^2]\)

\( L \), Length, \([m]\)

\( \bar{P} \), Time Averaged Pressure, \([Pa]\)

\( \Delta P \), Pressure Drop, \([Pa]\)

\( R \), Total Resistance, \([1/m]\)

\( R_c \), Cake Resistance, \([1/m]\)

\( R_m \), Mesh Resistance, \([1/m]\)

\( Re_D \), Reynolds Number Based on Diameter

\( t \), Time, \([s]\)

\( \bar{\mathbf{u}} \), Time Averaged Velocity Vector \( = [u,v,w] \), \([m/s]\)

\( u^* \), Dimensionless Velocity
\( u' \), Turbulent Velocity Fluctuation, [m/s]

\( U \), Bulk Fluid Velocity, [m/s]

\( x_i \), Position Vector \((= [x,y,z])\), [m]

\( \epsilon \), Eddy Dissipation Rate, \([m^2/s^3]\]

\( \mu \), Dynamic Viscosity of Fluid, \([Pa \cdot s]\]

\( \mu_t \), Turbulent (Eddy) Viscosity, \([Pa \cdot s]\]

\( \rho \), Fluid Density, \([kg/m^3]\]

\( \sigma_k \), Turbulence Model Constant

\( \sigma_\epsilon \), Turbulence Model Constant

\( \overline{\tau_{ij}} \), Time Averaged Laminar Stress Tensor, \([N/m^2]\)
List of Tables

<table>
<thead>
<tr>
<th>Table</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Experiment specifications.</td>
<td>20</td>
</tr>
<tr>
<td>2</td>
<td>Geometric properties of the tested mesh filters.</td>
<td>21</td>
</tr>
<tr>
<td>3</td>
<td>Results for the 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.</td>
<td>22</td>
</tr>
<tr>
<td>4</td>
<td>Parameters and discretization schemes chosen for CFD models.</td>
<td>23</td>
</tr>
<tr>
<td>5</td>
<td>Grid independence test for 158 [μm] mesh.</td>
<td>24</td>
</tr>
<tr>
<td>6</td>
<td>Grid independence study for 350 [μm] mesh.</td>
<td>25</td>
</tr>
<tr>
<td>7</td>
<td>Sensitivity analysis on the effects of inflow turbulence intensity on the pressure drop across mesh filter, using 5% turbulence intensity as a reference value.</td>
<td>26</td>
</tr>
<tr>
<td>8</td>
<td>Resistance coefficients for 158 and 350 [μm] mesh, obtained from CFD results.</td>
<td>27</td>
</tr>
</tbody>
</table>
**Table 1**: Experiment specifications.

<table>
<thead>
<tr>
<th>Component</th>
<th>Specification</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet Pipe</td>
<td>Length = 2.44m, Diameter = 0.041m</td>
</tr>
<tr>
<td>Outlet Pipe</td>
<td>Length = 1.52m, Diameter = 0.041m</td>
</tr>
<tr>
<td>Water Reservoir</td>
<td>80L</td>
</tr>
<tr>
<td>Velocity Range</td>
<td>0.3 - 1.5 [m/s]</td>
</tr>
<tr>
<td>Pump</td>
<td>Gould 1HP (745.7 W), centrifugal pump</td>
</tr>
<tr>
<td>VFD Controller</td>
<td>SMVector, 1.5HP (1118.55 W)</td>
</tr>
</tbody>
</table>
### Table 2: Geometric properties of the tested mesh filters.

<table>
<thead>
<tr>
<th>Nominal Pore Size [$\mu m$]</th>
<th>Thread Diameter [$\mu m$]</th>
<th>Open Area Percentage [%]</th>
</tr>
</thead>
<tbody>
<tr>
<td>158</td>
<td>145</td>
<td>26</td>
</tr>
<tr>
<td>350</td>
<td>250</td>
<td>34</td>
</tr>
</tbody>
</table>
Table 3: Results for the 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.

<table>
<thead>
<tr>
<th>Radial position [m]</th>
<th>Velocity [m/s]</th>
<th>Turbulent Kinetic Energy [m²/s²]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Coarse</td>
<td>Fine</td>
</tr>
<tr>
<td>0.00108</td>
<td>0.59613</td>
<td>0.59610</td>
</tr>
<tr>
<td>0.00538</td>
<td>0.58891</td>
<td>0.58890</td>
</tr>
<tr>
<td>0.00968</td>
<td>0.57349</td>
<td>0.57352</td>
</tr>
<tr>
<td>0.01399</td>
<td>0.53363</td>
<td>0.53365</td>
</tr>
<tr>
<td>0.01829</td>
<td>0.45509</td>
<td>0.45509</td>
</tr>
</tbody>
</table>
Table 4: Parameters and discretization schemes chosen for CFD models.

<table>
<thead>
<tr>
<th>Category</th>
<th>Setting</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>General</td>
<td>Solver</td>
<td>Pressure based, steady, absolute reference frame</td>
</tr>
<tr>
<td>Models</td>
<td>Energy</td>
<td>Isothermal</td>
</tr>
<tr>
<td></td>
<td>Viscous</td>
<td>Turbulent (k-ε with enhanced wall function)</td>
</tr>
<tr>
<td>Materials</td>
<td>Fluids</td>
<td>Water</td>
</tr>
<tr>
<td>Solution Methods</td>
<td>Pressure-Velocity Coupling Scheme</td>
<td>SIMPLE</td>
</tr>
<tr>
<td></td>
<td>Gradient</td>
<td>Least Squares Cell Based</td>
</tr>
<tr>
<td></td>
<td>Pressure</td>
<td>Second Order</td>
</tr>
<tr>
<td></td>
<td>Momentum and Turbulence Advection</td>
<td>Second Order Upwind</td>
</tr>
<tr>
<td>Residuals</td>
<td>Scaling</td>
<td>Local Scaling</td>
</tr>
<tr>
<td></td>
<td>Convergence Criteria</td>
<td>1.0 × 10⁻⁵</td>
</tr>
</tbody>
</table>
Table 5: Grid independence test for 158 [µm] mesh.

<table>
<thead>
<tr>
<th>Number of Control Volumes</th>
<th>Pressure Drop [Pa]</th>
<th>Percent Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>50002</td>
<td>7209.55</td>
</tr>
<tr>
<td>Medium</td>
<td>75485</td>
<td>7356.66</td>
</tr>
<tr>
<td>Fine</td>
<td>160185</td>
<td>7386.10</td>
</tr>
</tbody>
</table>
Table 6: Grid independence study for 350 [μm] mesh.

<table>
<thead>
<tr>
<th>Number of Control Volumes</th>
<th>Pressure Drop [Pa]</th>
<th>Percent Change</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>25251</td>
<td>1693.80</td>
</tr>
<tr>
<td>Medium</td>
<td>45178</td>
<td>1824.38</td>
</tr>
<tr>
<td>Fine</td>
<td>61428</td>
<td>1802.86</td>
</tr>
</tbody>
</table>
Table 7: Sensitivity analysis on the effects of inflow turbulence intensity on the pressure drop across mesh filter, using 5% turbulence intensity as a reference value.

<table>
<thead>
<tr>
<th>Turbulence Intensity</th>
<th>$P_{inlet} - P_{outlet}$</th>
<th>% Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5302.52</td>
<td>0.26</td>
</tr>
<tr>
<td>2</td>
<td>5304.6</td>
<td>0.22</td>
</tr>
<tr>
<td>3</td>
<td>5307.4</td>
<td>0.17</td>
</tr>
<tr>
<td>4</td>
<td>5310.8</td>
<td>0.10</td>
</tr>
<tr>
<td>5</td>
<td>5316.23</td>
<td>-</td>
</tr>
<tr>
<td>6</td>
<td>5320.87</td>
<td>0.09</td>
</tr>
<tr>
<td>7</td>
<td>5325.9</td>
<td>0.18</td>
</tr>
<tr>
<td>8</td>
<td>5331.25</td>
<td>0.28</td>
</tr>
<tr>
<td>9</td>
<td>5336.91</td>
<td>0.39</td>
</tr>
<tr>
<td>10</td>
<td>5340.47</td>
<td>0.46</td>
</tr>
</tbody>
</table>
Table 8: Resistance coefficients for 158 and 350 [$\mu$m] mesh, obtained from CFD results.

<table>
<thead>
<tr>
<th>Nominal Pore Size [$\mu$m]</th>
<th>a</th>
<th>b</th>
</tr>
</thead>
<tbody>
<tr>
<td>158</td>
<td>1032000</td>
<td>9306000</td>
</tr>
<tr>
<td>350</td>
<td>30900</td>
<td>3634000</td>
</tr>
</tbody>
</table>
List of Figures

1 Schematic diagram of the experimental setup. ................................. 29
2 Microscope images of meshes with nominal pore sizes of (a) 158 and (b) 350 µm. .................................................. 30
3 Diagrams of (a) the idealized mesh filter model with no geometric weave and (b) a cross-section of the computational domain with the relevant dimensions in terms of the thread diameter, D. ................................. 31
4 Normalized velocity profile at pipe outlet for varying inlet velocities. .......................... 32
5 Turbulence intensity profile at pipe outlet for varying inlet velocities. ................ 33
6 Plot of the experimental results for pressure drop as a function of bulk flow velocity in a pipe flow with filter insert, in comparison to numerical results for pressure drop in a representative filter element for mesh filters with nominal pore sizes of 158 and 350 µm. .................................................. 34
Figure 1: Schematic diagram of the experimental setup.
Figure 2: Microscope images of meshes with nominal pore sizes of (a) 158 and (b) 350 µm.
Figure 3: Diagrams of (a) the idealized mesh filter model with no geometric weave and (b) a cross-section of the computational domain with the relevant dimensions in terms of the thread diameter, $D$. 
Figure 4: Normalized velocity profile at pipe outlet for varying inlet velocities.
Figure 5: Turbulence intensity profile at pipe outlet for varying inlet velocities.
Figure 6: Plot of the experimental results for pressure drop as a function of bulk flow velocity in a pipe flow with filter insert, in comparison to numerical results for pressure drop in a representative filter element for mesh filters with nominal pore sizes of 158 and 350 µm.