International Journal of Recent Technology and Engineering (IJRTE)
ISSN: 2277-3878 (Online), Volume-8 Issue-6, March 2020

Experimental and Numerical Analysis of Fluid Flow in Microfibers

Madhusmita Ghadai, Subrat Kumar Barik

Abstract: ***The flow in small geometries is of interest in applications like bioanalysis systems, micro-valves and flow through porous media. The flow generally straight- forward to predict since the flow is stratified and so the geometries are clear-cut. However, once it involves flow through twin scale porous media, the flow gets harder to predict. The flow-through porous media are usually applied to areas like composites manufacturing, paper making and drying of ore pellets. The aim of this paper is to check flows in small geometries with numerical and experimental ways to realize an enlarged understanding of porous media flow. The optical technique best fitted to this sort of geometries is micro image velocimetry (µ-PIV) and numerical calculations are done with computational fluid dynamics (CFD). µ-PIV is employed to analyze the flow in channels with one fiber and with fiber arrays of various patterns and densities. The impact consistency has on flow fields in channels is investigated with CFD.***

Keywords: Computational fluid dynamics, flow through a porous medium, Microimage velocimeter, Micro Fiber.

I. INTRODUCTION

There are two states of fluid flow. When the flow is said to be laminar, it means that the flow is highly ordered and has smooth streamlines. A streamline is defined as a curve that is everywhere tangent to the instantaneous local velocity vector. When the flow has velocity fluctuations and disordered motion, the flow is said to be turbulent. Several parameters affect the transition from laminar to turbulent flow, for instance, the geometry, fluid velocity and material properties of the fluid. The experimental work of Osborne Reynolds in the 1880s led to the conclusion that the transition could be described as a ratio of the inertial forces to viscous forces in the fluid. [1].This ratio is known as the Reynolds number and is defined as Re = ρVL / μ

Where,

Manuscript received on February 10, 2020.
Revised Manuscript received on February 20, 2020.
Manuscript published on March 30, 2020.

* Correspondence Author

Madhusmita Ghadai*, Assistant Professor, GMR Institute of Technology and Management, Rajam, Srikakulam. Andhra Pradesh, India.
E-mail: madhusmita.g@gmrit.edu.in,

Subrat Kumar Barik, Assistant Professor, Mechanical Engineering, Sri Sivani College Of Engineering.Srikakulam, Andhra Pradesh, India.
E-mail:subrat.ssce@gmail.com,

ρ = density of the fluid,
V = velocity of the fluid,
μ = viscosity of fluid,
L = length or diameter of the fluid.

Reynolds number formula can be used in the problems to calculate the Velocity **(V)**, density (ρ), Viscosity (μ) and diameter **(L)** of the liquid.

In the case of flow-through micro geometries, the flow is laminar in almost every case because of the small scales involved. The flow can be explained by the equations of fluid motion. The continuity equation and the Navier-Stokes equation is defined as follows

$$\frac{\partial P}{\partial t} + \nabla.(\rho u) = 0$$

$$\rho\left(\frac{\partial u}{\partial t} + (u.\nabla)u\right) = -\nabla p + \rho\nabla^2 u$$

Solving the Navier-Stokes equation for anything except simple flow fields is not possible at this present time since it is a time-dependent, nonlinear, second-order partial differential equation. Since analytical solutions are not possible for more complex cases it is therefore of great interest to analyze these cases experimentally or numerically.

A. Porous media

The flow in porous media has been taken a great interest for a long time. Henry Darcy studied the filtering of drinking water in the city of Dijon as early as 1856. From the experimental observations, he derived a one-dimensional law for fluids propagating through a porous media. His law was theoretically derived and extended to several dimensions to take the form

$$Vs = -\frac{K}{\mu}\nabla p$$

Where Vs [m/s] is the superficial velocity (ratio between volumetric flow rate through the porous medium and the cross-sectional area in the flow direction) and K [m2] is the permeability tensor of the porous medium [2].

The law is valid as long as the Reynolds number is low enough to ensure laminar flow, the fluid is incompressible and Newtonian and the porous domain is stationary.

B. Optical measuring methods

Optical measuring methods have been used for several years. These kinds of methods are well suited for fluid mechanical problems since mechanical methods tend to disturb the flow. Among the first to use tracker particles to monitor flows was Ludwig Prandtl. At the beginning of the 1900s, he performed experiments on the flow around objects in a water tunnel where particles were introduced to the surface of the flow.

From these experiments, Prandtl was able to show flow phenomena qualitatively. With the development of technology in the last couple of decades, mostly the transition to digital recording and evaluation has given the optical measurement techniques the ability to give quantitative data such as velocities.

Particle image velocimetry or Laser speckle velocimetry as it was called early on started to develop in the late 70s and early 80s. In the early work of Roland Meynart, he showed that it was possible to make practical measurements on both laminar and turbulent flow of fluids with this method [3, 4]. The concept of applying this technique to sub-millimeter scaled geometries started developing in 1998 where the flow around an elliptical cylinder with a major diameter of 30 µm was investigated [5]. The flow-through rectangular networks of cylinders have been investigated with the PIV-method where different solid volume fractions and fiber radii were looked at [6, 7].

C. Computational Fluid Dynamics

Solving fluid-related problems numerically with Computational Fluid Dynamics has become a standard industrial tool. Solving the Reynolds-Averaged Navier-Stokes (RANS) equations with the help of computers is considered a good method for solving problems that are too complex to solve analytically. It is important to understand that the solutions obtained from CFD will always be approximate because a CFD model is always a simplification of reality.

II. METHOD

A. Particle Image Velocimetry

Particle Image Velocimetry is a method that allows complex instantaneous velocity fields to be measured [13]. In a typical PIV-setup, the flow is seeded with tracer particles and a plane is illuminated two times or more in a short time. The emitted light from the particles is then recorded by a camera. The recordings are evaluated on a computer by dividing the images into smaller subareas and comparing the particle placement in these sub-areas.

The assumption is made that the particles move linearly within the subarea when the time between images Δt is sufficiently small. By correlating the placement of the particles in sequential images, both the magnitude of the velocity and the direction can be evaluated, see Fig. 2.1.

When applying the PIV technique to micro geometries some adjustments must be made. The investigated domain is imaged through a microscope before it is captured by the camera. Since the illumination of a single plane is difficult to achieve in these kinds of geometries the entire volume is illuminated by the light source. The limitation in measurement depth is instead decided by the focus of the microscope.

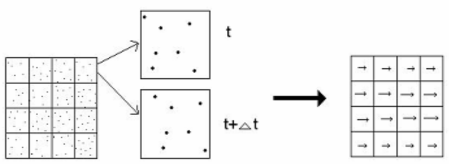


Figure 2.1.The basic principle behind cross-correlation

B. Seeding Particles

The seeding particles have a big impact on how the results of an experiment will turn out. Both the size of the particles and the particle concentration are parameters that should be chosen in such a way that they match the studied geometry and flow velocity. The particles should be small enough to follow the flow in a good way but not so small that they will be affected by random disturbances in their movement which is known as Brownian motion. The relative error that occurs from Brownian motion can be estimated by

$$\varepsilon_b = \frac{1}{u}\sqrt{\frac{2D}{\Delta t}}$$

Where D [m2=s] is Einstein's diffusion coefficient and Δt [s] is the measurement interval [5].

The particle concentration should be kept as low as possible to keep a good signal-to-noise ratio [14]. This is because the number of particles that move outside the investigated plane is lower and their emitted light will not add as much noise to the pictures obtained. Since low particle concentrations can lead to insufficient data to perform correlations between two consecutive images the "' Sum of Correlations method"' was used. This method will sum up an arbitrary number of correlations before the velocity field is calculated [15]. Paraffin oil was used as the fluid in all experiments because it has a refraction index.

C. Experimental setup

Close to that of the glass walls of the channels. It is important to have a homogenous particle distribution in the fluid. If particles start to clump together they will disturb the velocity field obtained from the experiments. When mixing the particles into the paraffin oil special care has to be taken or a lot of air will be added to the fluid. Since the air bubbles generally are larger than the tracer particles they will have a significant negative effect on the results. The first method was simple to apply the particles to the surface of the container that held the fluid and then stirs. With the stirring method, a lot of air is trapped inside the fluid and a lot of particles adhere to the walls. The method that seemed to give the best fluid was to use a sonic bath for the mixing. Placing the container with the fluid and tracker particles in the sonic bath where small vibrations handled the mixing gave a very homogenous distribution of particles in the fluid. The procedure does, however, add some air to the mixture and this had to be dealt with in some way. The solution was to put the container which held the mixture in a vacuum pump to get rid of all the residual air.

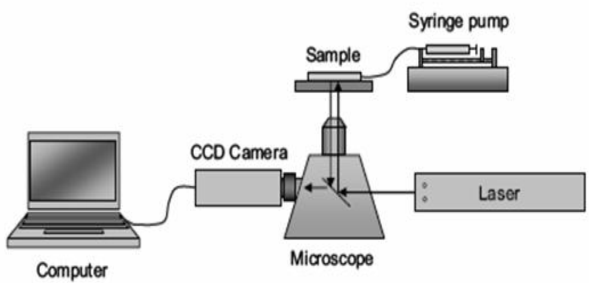


Figure 2.2.Experimental setup

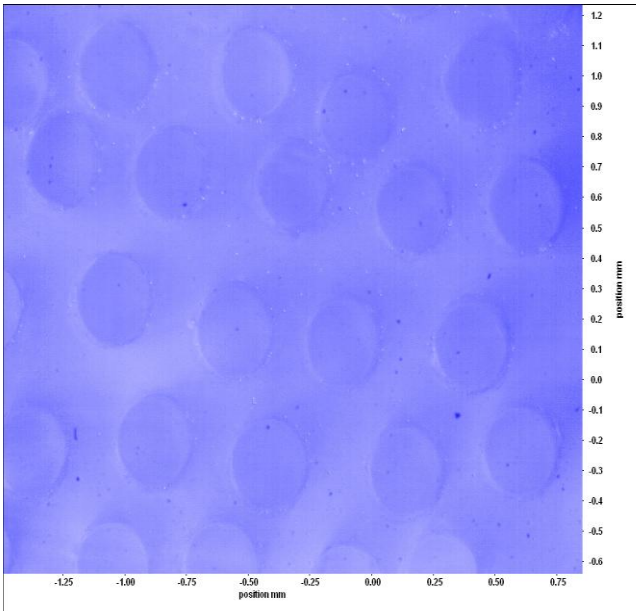


Figure 2.3. The optical view of a fiber bundle

D. Flow around Single Fiber

The geometry for the numerical model of the flow around one fiber was created in ANSYS Workbench and the unstructured mesh was created in ICEM CFD. The unstructured mesh had 737k tetrahedral elements. A local refinement near the cylinder was applied to give increased accuracy near the cylinder surface, see Fig. 2.4.

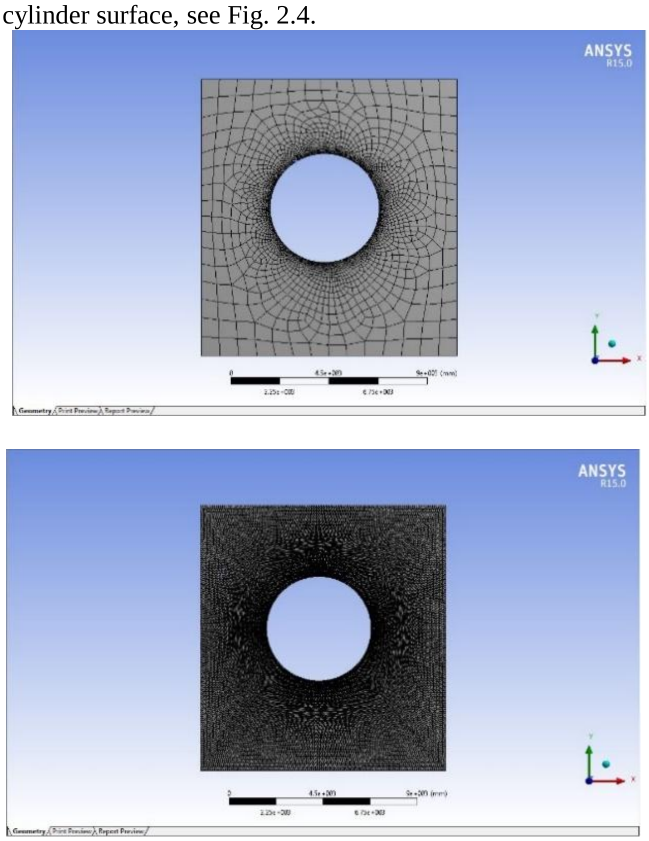
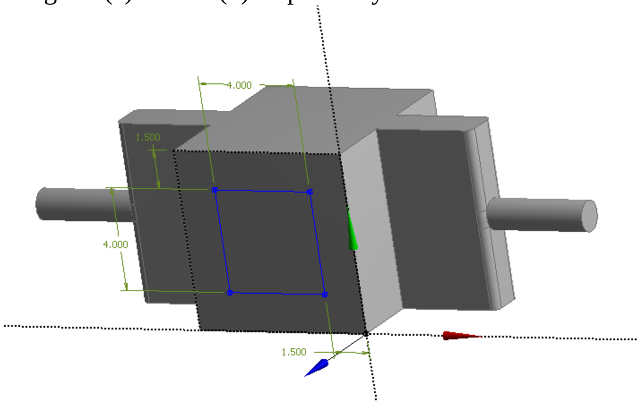


Figure 2.4. Mesh

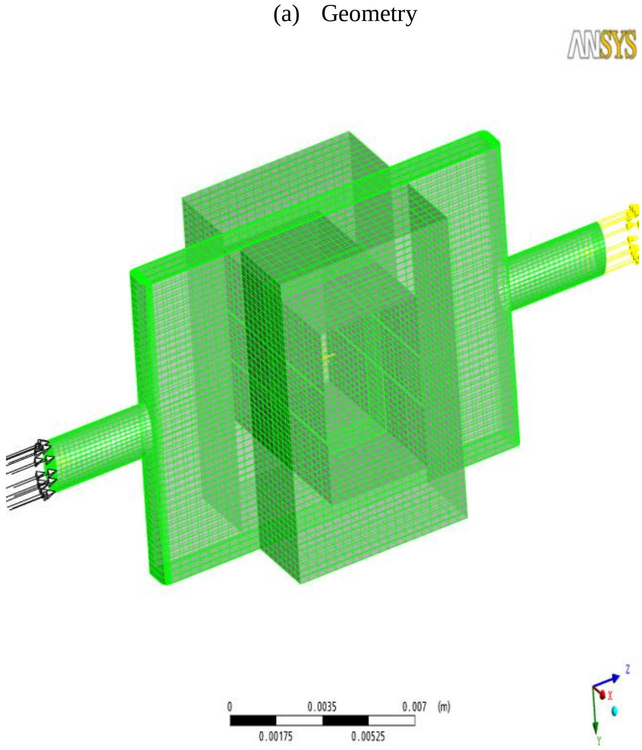
The inlet boundary condition is set as a plug profile with velocity obtained from experimental data. The cylinder and the top and bottom walls are modeled with a no-slip boundary condition and the front and back walls are modeled as symmetry planes to remove wall effects. The outlet uses an average static pressure of 0Pa. The root means square (RMS) residual targets were set to 1e- 6 which is very tight convergence suitable even for geometrically sensitive problems [17].

E. Flow-through Fiber Bundle

The geometry for the channel with the porous material was the same as the geometry of the channel used in the experiments with the fiber bundles. The geometry was created and a block-structured mesh was created which had 1470k nodes. The block structure for the mesh was built up by an On-grid which starts at the inlet and goes all the way to the outlet and the areas around the On-grid were meshed as homogenous as possible. The geometry and mesh can be seen in Fig. 2.5(a) and 2.5(b) respectively.



(a) Geometry



(b)Mesh

Figure 2.5: Geometry and mesh for numerical model

The velocity at the inlet was set to a plug profile with a velocity corresponding to the inlet velocity of the experiments.

A plug profile is not a very realistic assumption but it was considered a reasonable approximation since a fully developed profile will be obtained very soon in the tiny capillary leading to the cavity.

All walls were modeled with a no-slip boundary condition. The outlet was set to use an average static pressure of 0Pa. The fiber bundle was modeled as a porous subdomain with constant permeability. This approach was chosen because it's easier to model a complex 3D structure this way than to model every fiber in the fiber bundle. The permeability for hexagonal fiber arrays can be calculated as

$$K = \frac{8}{C}\frac{(1-f)^3}{f^2}R^2$$

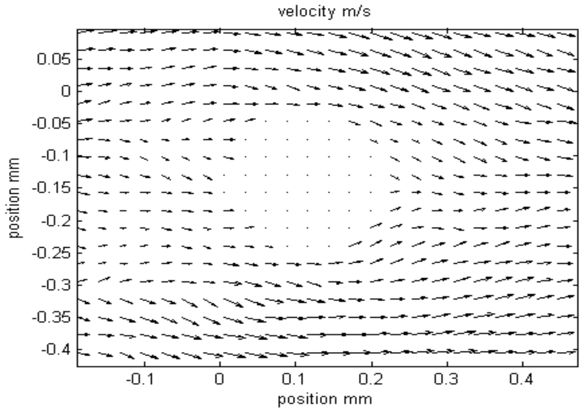
$$K = C\left(\sqrt{\frac{f_{max}}{f}} - 1\right)^{\frac{5}{2}}R^2$$

Where f is the fiber bundle volume fraction, C is a constant close to unity that is dependent on the actual fiber arrangement and R [m] is the fiber radius [18].

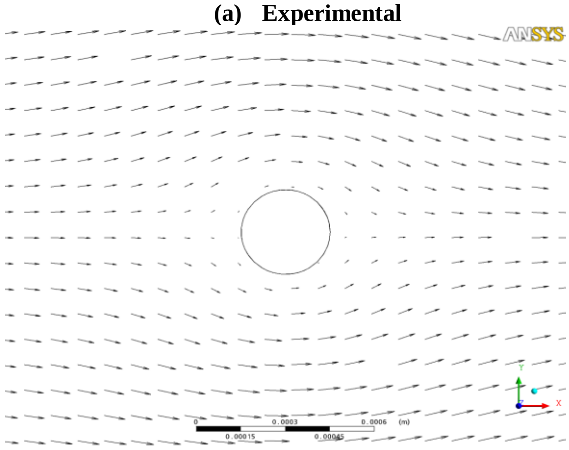
III. RESULTS

A. Flow around the single fiber

The velocity field obtained from PIV and the one obtained from CFD can be seen in Fig. 3.1(a) and 3.1(b) respectively. The results look very similar and the flow is much like what one would expect for flows with low Reynolds numbers. With very low upstream velocities the fluid completely wraps the cylinder and the flow going above the cylinder and the flow going beneath it will meet behind the cylinder in an ordered manner. For RE≥10 there will be some separation that starts occurring behind the cylinder but in this case, RE≤1 and hence no separation will occur.



(a) Experimental



(b) Numerical

Figure 3.1: Velocity fields

B. Flow-through fiber bundles

Since the PIV seemed to handle the single fiber well, a network of fibers placed as a rectangular array was investigated. Fig. 3.2 shows the obtained velocity field. There seems to be very little movement between the different rows of fibers. It can also be noted that the velocity in the x-direction increases in the necks between the fibers which is clearly shown in Fig. 3.3.

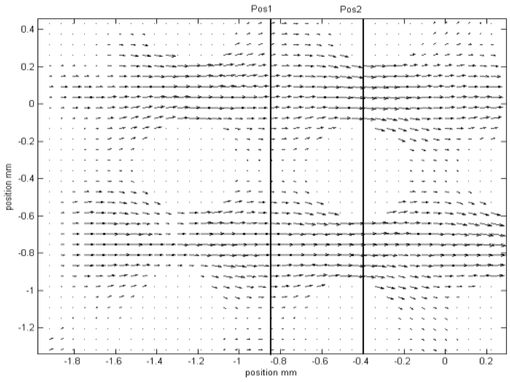


Figure 3.2: Velocity field for flow through rectangular fiber array

The trouble with air getting suspended inside the channel did give a unique opportunity to see how disturbances in the array affect the flow. In Fig. 3.4 there is an air bubble trapped between two fibers. In the obtained velocity field some of the flow moving over the bubble is forced to go above the next fiber row as seen in Fig. 3.5.

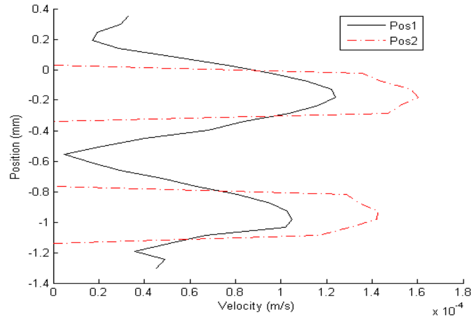


Figure 3.3: velocity profiles for flow through rectangular fiber array

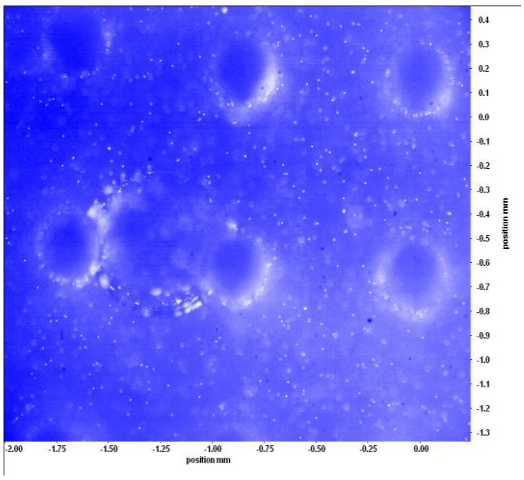


Figure 3.4: Air bubble trapped between fibers

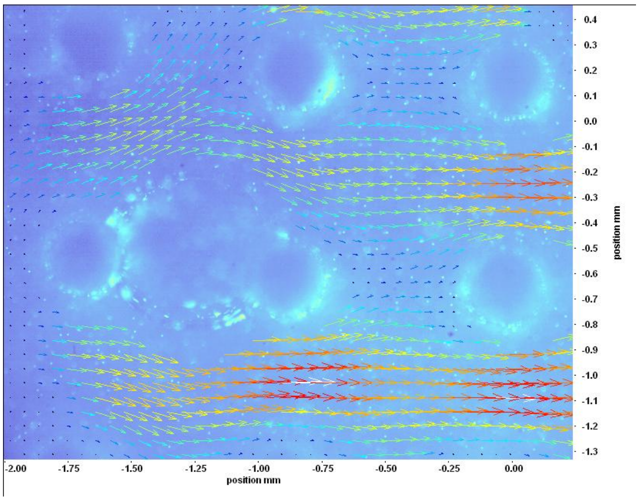


Figure 3.5: Velocity field of the array with air bubble

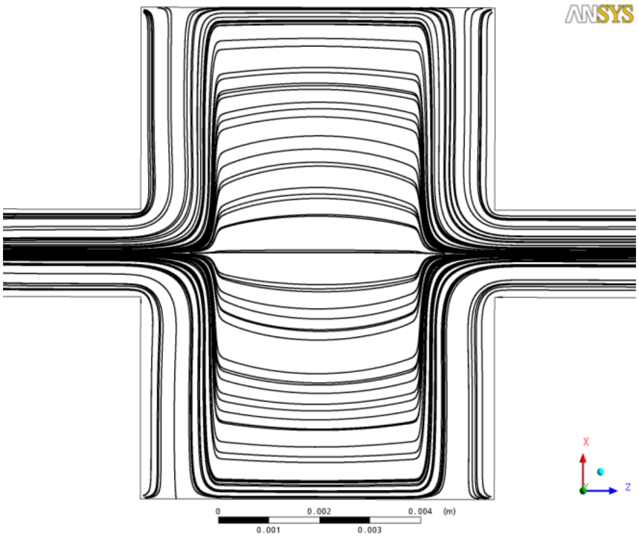


Figure 3.7: Streamlines for velocity in the plane perpendicular to the measurement plane

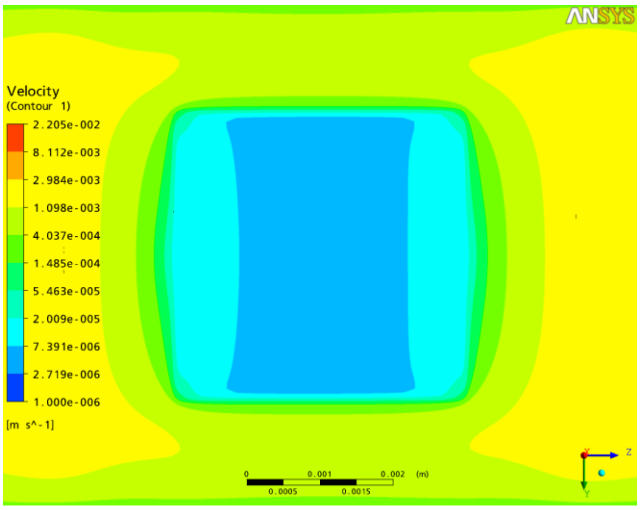


Figure 3.8: Contour plot at the porous domain

IV. DISCUSSION

Microparticle image velocimetry and computational fluid dynamics were used to investigate flows in simple sub-millimeter geometries. Since reasonable results were obtained the more complex case of porous media was looked at. The porous material in the experiments consisted of fiber arrays in different alignments and solid volume fractions. The obtained flow fields from PIV shows that the technique can be used even in very dense arrays where the velocities are very slow compared to the bulk flow. Although all the studies performed with the PIV was considered stationary, the results obtained indicate that it should be possible to monitor flows in transient applications such as flow fronts in filling processes. It should also be possible to mix larger particles in the fluid to see how they hinder the fluids propagation through the porous media. When looking at transient events there will be little to no room for optimizing settings after the measurement has started and parameters such as laser power, microscope focus and time between laser pulses all have a great impact on the final results. One way to handle this would be to run a stationary case first to evaluate the different parameters and selecting optimal settings for them. Numerical simulations were carried out with the porous media modeled with constant permeability according to analytical formulas. The results showed that the permeability of the porous material affected the flow field both inside the fiber bundle but also the flow around it. This kind of simulations could be used with more complex models for permeability or porosity to get physical properties such as velocities or temperatures. It should also be possible to use this method to run transient simulations where flow fronts could be examined over time. Another interesting aspect would be to add particles to the fluid and use particle tracking to monitor their movement through the flow domain.

V. CONCLUSION AND FUTURE SCOPE

The experimental and numerical analysis that was applied to the samples of microfibers had given a good result.

By visualizing the result images and the output came from both numerical and experimental analysis the fluid flow through the microfibers are easily understood. So many fluid flow mathematical models can be established by considering the results. It can be also used for complex structures and complicated modeling in the future.

REFERENCES

1. Y. A. Cengel and J. M. Cimbala, Fluid Mechanics: Fundamentals and Applications. McGraw-Hill, 2006.
2. J. Bear, Dynamics of Fluids in Porous Media. American Elsevier Publishing Company, 1972.
3. R. Meynart, “Equal velocity fringes in a rayleigh-benard flow by a speckle method,” Appl. Opt., vol. 19, no. 9, p. 1385, 1980.
4. R. Meynart, “Speckle velocimetry study of vortex pairing in a low-Re unexcited jet,” Physics of Fluids, vol. 26, pp. 2074–2079, 1983.
5. J. G. Santiago, S. T. Wereley, C. D. Meinhart, D. J. Beebe, and R. J. Adrian, “A particle image velocimetry system for microfluidics,” Experiments in Fluids, vol. 25, pp. 316–31, 1998.
6. M. Agelinchaab, M. F. Tachie, and D. W. Ruth, “Velocity measurement of flow through a model three-dimensional porous medium,” Physics of Fluids, vol. 18, no. 1, p. 017105, 2006.
7. W. H. Zhong, I. G. Currie, and D. F. James, “Creeping flow through a model fibrous porous medium,” Experiments in Fluids, vol. 40, pp. 119–126, Jan. 2006.
8. ERCOFTAC Special Interest Group, Quality and Trust in Industrial CFD: Best Practice Guidelines. 2000.
9. J. H. Ferziger and M. Peric, Computational Methods for Fluid Dynamics. Springer-Verlag, 2002.

10. M. Nordlund, T. S. Lundström, V. Frishfelds, and A. Jakovics, “Permeability network model for non-crimp fabrics,” Composites Part A, vol. 37A, pp. 826–835, 2006.
11. X. Chen and T. D. Papathanasiou, “Micro-scale modeling of axial flow through unidirectional disordered fiber arrays,” Composites Science and Technology, vol. 67, pp. 1286–1293, 2007.
12. X. Chen and T. D. Papathanasiou, “The transverse permeability of disordered fiber arrays: A statistical correlation in terms of the mean interfiber spacing,” Transport in Porous Media, vol. 71, pp. 233–251, 2008.
13. M. Raffel, C.Willert, and J. Kompenhans, Particle Image Velocimetry: A Practical Guide. Springer-Verlag, 1998.
14. C. D. Meinhart, S. T. Wereley, and M. H. B. Gray, “Volume illumination for two-dimensional particle image velocimetry,” Meas. Sci. Technol., vol. 11, pp. 809–814, 2000.
15. LaVision GmbH, Davis Flowmaster Software manual. LaVision GmbH, Göttingen, Germany, 2005.
16. M. Nordlund, Permeability Modelling and Particle Deposition Mechanisms Related to Advanced Composites Manufacturing. Luleå University of Technology, PhD thesis, 2006.
17. ANSYS, CFX 11.0 Documentation. Ansys Inc, 2006.
18. B. R. Gebart, “Permeability of unidirectional reinforcements for rtm,” Journal of Composite Materials, vol. 26, no. 8, pp. 1100–1133, 1992.

AUTHORS PROFILE

Madhusmita Ghadai, Assistant Professor, GMR Institute of Technology and Management, Rajam, Srikakulam. Andhrapradesh, madhusmita.g@gmrit.edu.in, Research Scholar in Centurion University of Technology and Management. Specialized in Fluid mechanics, Hydrology and Water Resource. Good in using GIS, Hydrological Modeling, Water quality modeling. Used to use STADPRO, ARC GIS, AUTOCAD 3D, SPSS. Guided Five UG projects.

Subrat Kumar Barik, Assistant Professor, Mechanical Engineering, Sri Sivani College Of Engineering.Srikakulam, Andhrapradesh. subrat.ssce@gmail.com, Research Scholar in Centurion University Of Technology and Management. Specialized in Production, Manufacturing, and Tool Design. Having A good Knowledge in CNC and Master-CAM. Used to use AUTOCAD, SOLID WORKS, CATIA and ANSYS. Guided Seven UG projects. Published TWO E-Books.