Journal of Basrah Researches ((Sciences)) Vol. (45). No. 2 (2019)

Available online at: http://brsj.cepsbasra.edu.iq

ISSN -1817 -2695

Received 15-5-2019, Accepted 8-8-2019

NUMERICAL INVESTIGATION OF FLUID FLOW IN CAPILLARY TUBES

Amir Hussein Ali*

University of Basrah, College of Education for Pure Science, Department of Physics.

***Corresponding author:**

E-mail address: amirss_3000@yahoo.com

Abstract

This study was interested with fluid flow through narrow tubes. Finite volume method has been used as a numerical technique by the aid of computer to achieve the study. Cylindrical narrow tubes, represented as case A and case B have been used. Case A was a tube of radius (2 cm) and length (20 cm) while case B which has been used to investigate the effect of narrow tube on fluid flow was a tube of radius (2 mm) and length (20 mm). The initial conditions and boundary conditions for both cases were similar. Due to its widespread applications, air has been used as fluid material for the both cases. No slip condition at the wall has been used and viscous fluid flow was considered for both cases. Different parameters have been investigated for the two cases and have been compared to each other such as velocity magnitude, dynamic pressure, static pressure and wall shear stress. Case A has Reynold's number (Re) higher than case B. The velocity profiles of fully developed flow were compared with the parabolic velocity profile. Clear difference between the two cases A and B has been shown because of the narrowness of case B. The maximum velocity was at the tube centerline and zero at the tubes wall because of no slip condition was applied. Due to the differences in the diameters of the two cases, the results showed apparent differences between the two cases for velocity magnitude, dynamic pressure, static pressure and wall shear stress as well. The results of this study could be important for applications which include air flow in narrow tubes.

Keywords: Finite Volume methods, fluid flow**,** Capillary tubes.

1. Introduction

In fluid flow situation, the foundations are the conservation laws. These laws are based on classical mechanics and have been used to treat with fluid dynamics problems. Conservation of mass, conservation of linear momentum and conservation of energy could be written in integral or differential form. Furthermore the conservation laws could be applied to a region of the fluid flow called control volume. Also the integral form is used to describe the change of mass, momentum and energy within the control volume whereas differential formulations of the conservation laws apply Stoke's theorem at a point within the fluid flow.

2. Momentum Conservation Equation

Conservation of momentum in an inertial reference frame is described by [1]:

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla.(\rho\vec{v}\vec{v}) = -\nabla p + \nabla(\bar{\bar{\tau}}) + \rho\vec{g} + \vec{F} \qquad \text{......... (1)}$$

Where $\vec{v}$ is the velocity, ρ is the density, p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor, $\rho\vec{g}$ and $\vec{F}$ are the gravitational body force and external body force respectively. The stress tensor $\bar{\bar{\tau}}$ for Newtonian fluids is given by [1]:

$$\bar{\bar{\tau}} = \mu[(\nabla\vec{v} + \nabla\vec{v}^T) - \frac{2}{3}\nabla.\vec{v}\, I] \qquad \text{...... (2)}$$

where μ is the molecular viscosity, I is the unit tensor, and the second term on the right hand side is the effect of volume dilation.

3. Continuity Equation

The continuity equation states that the rate at which mass enters a system is equals to the rate at which mass leaves the system plus the accumulation of mass within the system. The equation for conservation of mass, or continuity equation, can be written as follow [1]:

$$\frac{\partial\rho}{\partial t} + \nabla.(\rho\vec{v}) = S_m \qquad \text{...... (3)}$$

The source S_m is the mass added to the continuous phase from the dispersed second phase (e.g., due to vaporization of liquid droplets) and any user-defined sources.

If the fluid is an incompressible flow (ρ is constant), and steady state, the mass continuity equation could simplify to the following equation [1]:

$$\nabla.\vec{v} = 0 \qquad \text{....... (4)}$$

From Reynold's number, for pipe flows of various diameters, it was observed that there is a critical Reynold's number (2300), below this value; pipe flow is laminar whereas up this value, pipe flow is turbulent [2].

$$Re_{critical} = 2300 = \frac{\rho v D}{\mu} \qquad \text{......... (5)}$$

For the air:

$\mu_{air} = 1.80 \times 10^{-5}$ kg/ m.s , $\rho_{air} = 1.205$ kg/m^3

4. Laminar Flow in Pipes

Flow in pipes is laminar for Re $\leq$ 2300.
The velocity profile in fully developed laminar flow in a pipe is represented by equations (6-8) as follow [3]:

$$v(r) = \frac{R^2}{4\mu}\left(\frac{dp}{dx}\right)\left(1 - \frac{r^2}{R^2}\right) \qquad \text{........ (6)}$$

$$v(r) = 2v_{avg}\left(1 - \frac{r^2}{R^2}\right) \qquad \text{........ (7)}$$

$$v_{max} = 2v_{avg} \qquad \text{........ (8)}$$

where R is the pipe radius and r is a distance from the centerline toward the wall of the tube. μ is the viscosity. Also v_{avg} is the average velocity. Therefore, the velocity profile, $v(r)$ in fully developed laminar flow in a pipe is parabolic with a maximum value (v_{max}) at the centerline (r=0) and zero at the pipe wall, (r=R).

5. Navier-Stokes equation

The Navier-Stokes equation for incompressible, Newtonian (with a constant viscosity) fluid is given as [4]:

$$\rho\left(\frac{\partial\vec{v}}{\partial t} + \vec{v}\,.\,\nabla\,\vec{v}\right) = -\nabla p + \mu\,\nabla^2\,\vec{v} + \rho\vec{g} \qquad \text{........ (9)}$$

6. Knudsen number

The Knudsen number, *Kn*, relates the mean free path, λ, to the system dimension, $Kn=\lambda/L$. It should be noted that the average distance between collisions i.e. the mean free path, in air at 1 atm and room temperature is about to 80 nm. Corrections of Navier-stokes equation and boundary conditions are required for Knudsen numbers larger than ~0.02 [5].

The advancements in micro-and nano-techniques have led researchers to invest unprecedented levels of attention in the science and technology of micro- and nano-scale devices and their related applications. Micro-fluid has been an important research area for the last twenty years. Recently, most of the researches were emphasis on the finite volume numerical methods. To deal with the fluid flow and heat transfer problems, the modern and powerful technique is the computer-based simulation. The technique is very powerful and spans for a wide range of industrial application fields. [6-11]. In literature, fluid flow in different conditions has great attention.

G. Kaoullas and G. C. Georgiou [12] derived analytical solutions for the steady incompressible Newtonian poiseuille flow in various geometries. For the axisymmetric problem there were two distinct flow regimes corresponding to no-slip and slip defined by a critical value of the pressure gradient, which depends on the slip yield stress. They found that annular problem was more complicated with three distinct flow regimes defined by two critical values of the pressure gradient. The Newtonian Poiseuille flow is considered for various geometries, under the assumption that wall slip occurs above a critical value of the wall shear stress known as, the slip yield stress. Two critical pressure gradients characterize the annular and rectangular Poiseuille flows. Below the first critical value no-slip occurs while above the second one, slip occurs at all walls. In the intermediate regime for the annular problem, slip occurs only at the inner-wall, while for the rectangular problem, there are two intermediate regimes for which there are no analytical solutions. In the first regime slip occurs only in the middle sections of the wider walls and in the second one partial slip also occurs along the narrower walls. S. Nallapua and G. Radhakrishnamachya [13] investigated two-fluid model for the flow of a Jeffrey fluid in tubes of small diameters in the presence of a magnetic field. They assumed that the core region consists of a Jeffrey fluid and a Newtonian fluid in the peripheral region. Analytical expressions for velocity, flow flux, effective viscosity, core hematocrit and mean hematocrit have been derived. The effects of various relevant parameters on these flow variables have been studied. They observed that the effective viscosity decreases with

Jeffrey parameter but increases with tube radius. In addition, the core hematocrit decreases with Jeffrey parameter and tube radius.

7. Procedure

In this study, air flow has been investigated through cylindrical narrow tubes, represented as case A and case B. Case A is a tube of radius (2 cm) and length (20 cm) while case B which has been used for comparison was a tube of radius (2 mm) and length (20 mm). Case A has Reynold's number (Re) higher than case B. Furthermore both cases have Reynold's number higher than $(Re_{critical})$ .Figure (1.a.) is a prototype to represents both cases, A and B. The air is interred the tube from face 1 and leave from face 3. The initial conditions and boundary conditions for both cases were similar. Face 1 represents velocity inlet for the air flow, face 3 represents the outflow whereas face 2 is the surrounding circular cylindrical wall of the tube. The velocity of the air at the face 1 of the tube has been set at (50 m/s) for both cases. In order to achieve good results, fine meshing has been used for both cases. The calculations were performed under different Reynold's number for both cases. No slip at the wall has been used and viscous fluid flow has been adopted. Finite volume method has been used as a numerical technique by the aid of computer to solve momentum conservation equation and continuity equation. Furthermore in the finite volume methods it is quite easy to implement a variety of boundary conditions, since the unknown variables are evaluated at the centre of the volume elements, not at their boundary faces. Many parameters have been investigated for the two cases such as contour of velocity magnitude, contour of dynamic pressure, contour of static pressure and shear stress.
Figure (1.b.) is the sketch of the cylindrical tube dimensions of this study.

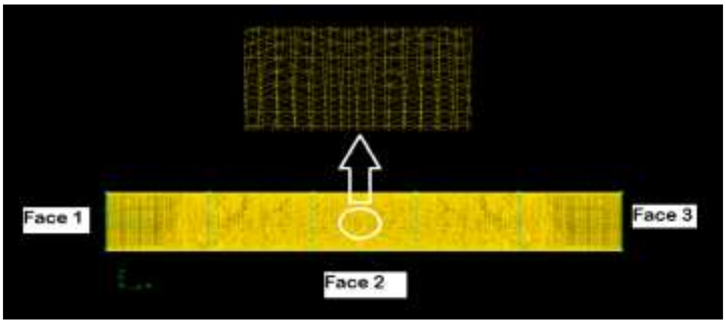

Fig.1.a. Prototype of the cylindrical tube showing the meshing type and quality.

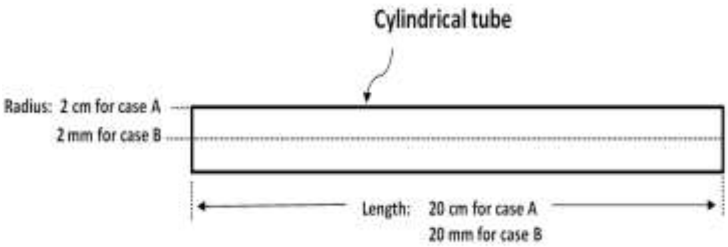

Fig.1.b. Sketch of the cylindrical tube dimensions.

8. Results and discussion

In fluid flow through a tube, different parameters and conditions could be affecting the physical properties of the fluid moving inside the tube. Furthermore the kind of fluid (gas or liquid) has its own properties. The tube dimensions have great attention in different studies. Recently, fluid flow in narrow tube could be more important because of wide variety of engineering applications such as flow of blood through narrow tubes, Venturi effect and Pitot tubes. This study focused on the effect of tube dimensions on the air flow properties. Two cases have been adopted, case A, and case B. Case A was, a tube of radius (2 cm) and length (20 cm) while case B which has been used to investigate the effect of narrow tube on fluid flow was a tube of radius (2 mm) and length (20 mm). The two cases have been simulated by solving the momentum conservation equation, (eq. 1) and continuity equation, (eq. 3). It should be noted that the result analysis will be at the outlet of the tube after air development i.e. at face (3) in figure (1.a). Also the air flow is in the x-axis. Because of the outlet (face 3) is circle and symmetric, hence, the radius of the circle (R) could be y-axis or z-axis. Figure (2) illustrates contour of velocity magnitude of the air flow for case A. The velocity profile is developed with zero magnitude at the tube surrounding surface (face 2) because of viscous and no slip conditions. On the outlet surface (face 3), the velocity increased gradually toward the center of the tube. At the center of the tube, the velocity reaches the maximum value as illustrated in Figure (3). The velocity profile from zero regions at the face 2 until the center of tube has been extracted from the simulation and illustrated in figure (4). In case (A), the effect of the wall on the shape and the magnitude of air velocity are decreased because the center of the tube is far from the wall. Hence the front of the parabolic profile of the velocity became wider and the flow is turbulent. For comparison, this real simulated velocity profile has been compared with the laminar velocity profile of equation (7) as illustrated in figure (5).

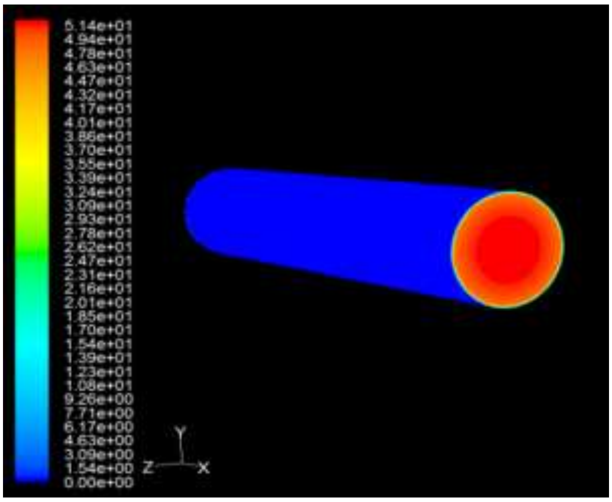

Fig.2. Contour of velocity magnitude of the air flow for case A.

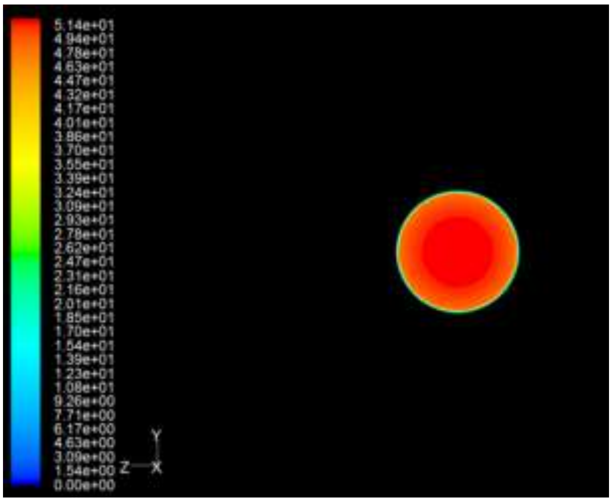

Fig.3. Contour of velocity magnitude of the air flow for case A at the outlet surface (face 3).

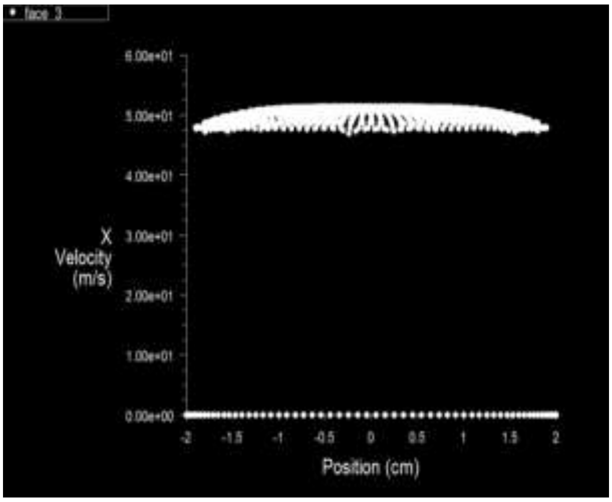

Fig.4. The real simulation velocity profile of case A.

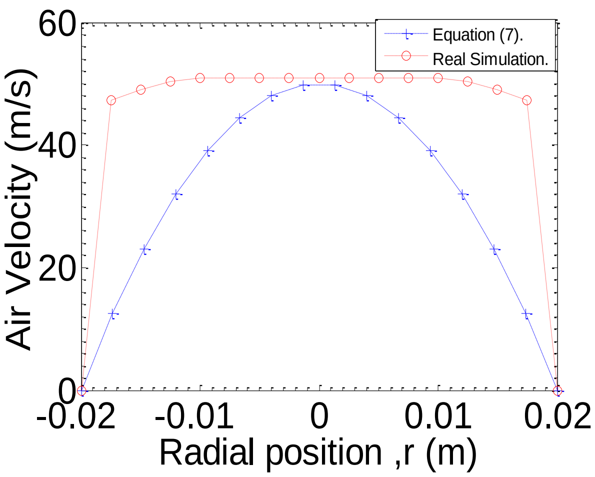

Fig.5. Velocity profile comparison between real simulation of case A and equation (7).

The investigation showed clear difference between the velocity profile of equation (7) and the real simulation.

In many applications, the pipes are not long enough to allow the attainment of fully developed flow. To do so the radius of the tube is reduced and investigated.

As the tube become narrower as the effect of the wall on the air velocity became more effective. Hence in case (B) where the tube is narrower, the magnitude of the velocity was increased at the centerline of the tube as shown in figures (6, 7). Furthermore the shape of the velocity profile in real simulation of turbulent flow is curved to take approximately parabolic shape as shown in figure (8). For comparison purpose, this real simulated velocity profile of case B has been compared with the perfect velocity profile of equation (7) as illustrated in figure (9). There is clear approach between real simulated velocity profile of case B and the perfect velocity profile of equation (7) compared to case A situation. Also real simulated velocity profile of case B became higher and above the velocity profile of equation (7). The velocity profile difference could be referred to the difference of Reynold's number between case A and case B. In literature, important information had been shown in this regard [14].

The air velocity is determined by the dynamic pressure and the static pressure. Hence, increasing the velocity magnitude in case (B) is reasonably accepted because of increasing the dynamic pressure on the expense of static pressure in case (B) compared to case (A) as illustrated in figures (10-13). It should be noted that the ratio of the length to radius of the tubes are constant for both cases. In literature, it was shown that the flow regime is controlled by the length-to-diameter ratio [15].

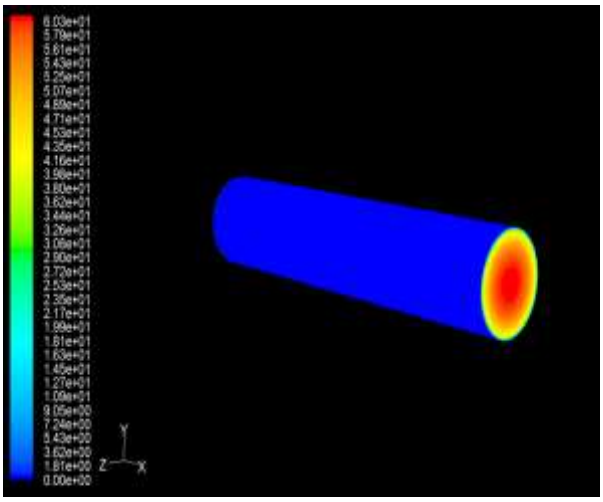

Fig. (6). Contour of velocity magnitude of the air flow for case B.

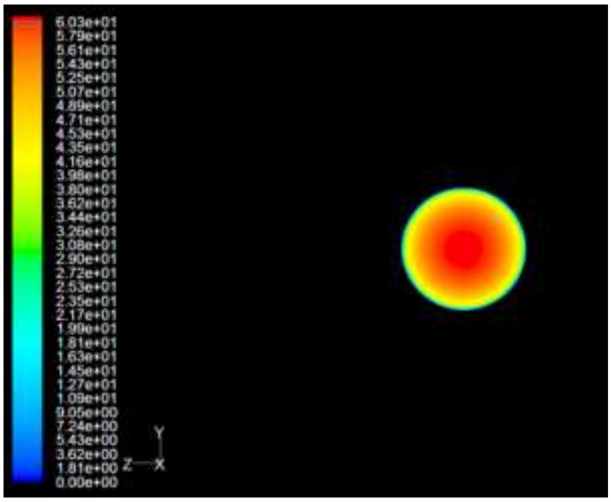

Fig.7. Contour of velocity magnitude of the air flow for case B at the outlet surface (face 3).

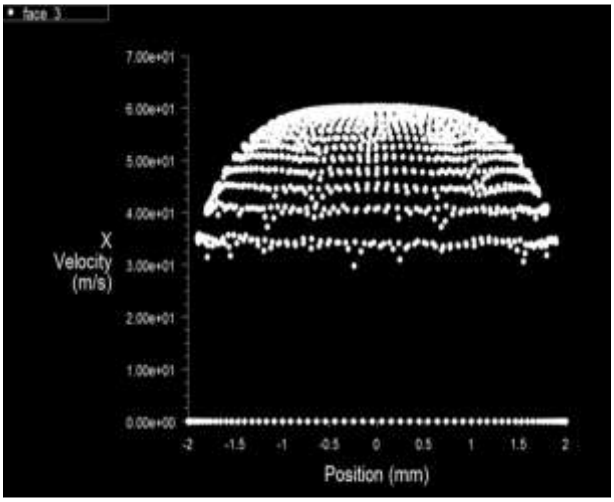

Fig.8. The real simulation velocity profile of case B.

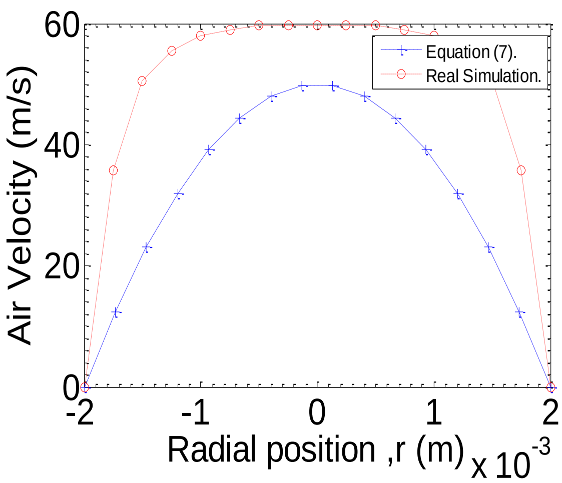

Fig.9. Velocity profile comparison between real simulation of case B and equation (7).

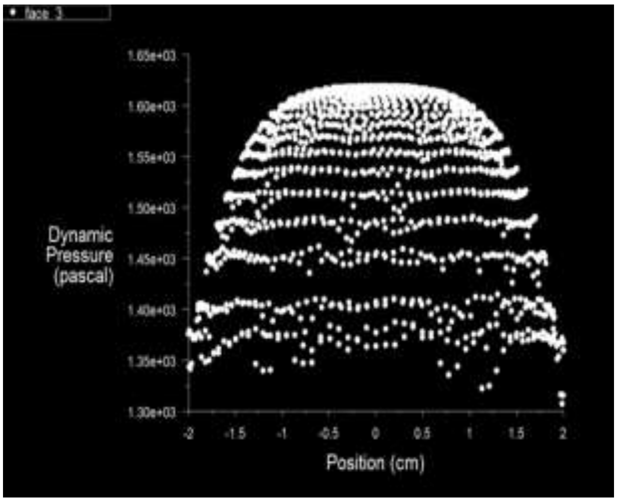

Fig.10. The real simulation dynamic pressure profile of case A.

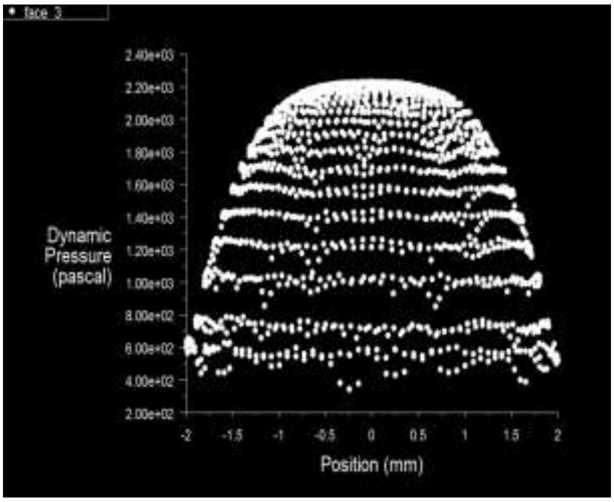

Fig.11. The real simulation dynamic pressure profile of case B.

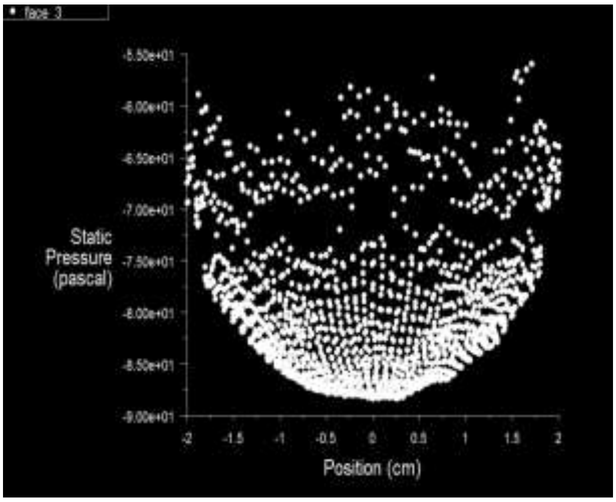

Fig.12. The real simulation static pressure of case A.

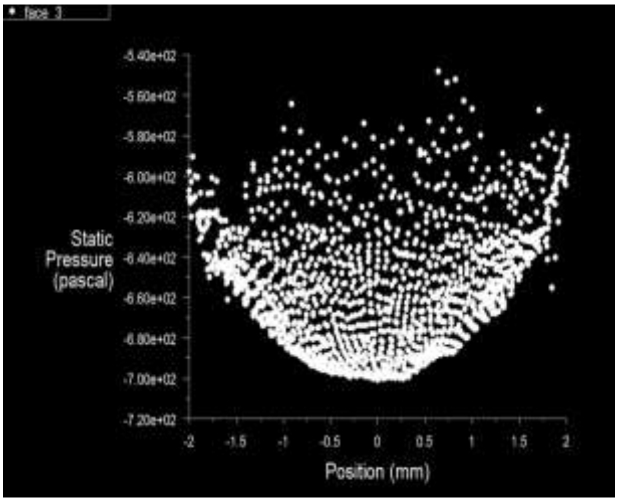

Fig.13. The real simulation static pressure of case B.

In real fluids (liquids or gases) moving along a solid boundary will incur a shear stress at that boundary. The no-slip condition dictates that the speed of the fluid at the boundary is stationary i.e., zero fluid velocity. Higher from the stationary velocity condition, the flow speed must equal that of the fluid. The region between these two points is named the boundary layer. In Newtonian fluids, the shear stress is proportional to the strain rate in the fluid, and the viscosity is the constant of proportionality.

The influence of the tube narrowness on the wall shear stress has been investigated. As mentioned earlier, the velocity of the air was increased in narrow tube. The affect of the velocity magnitude is clearly appeared on the wall shear stress investigations. The value of wall shear stress of case (B) is ten times that of case (A) as illustrated in figures (14, 15).

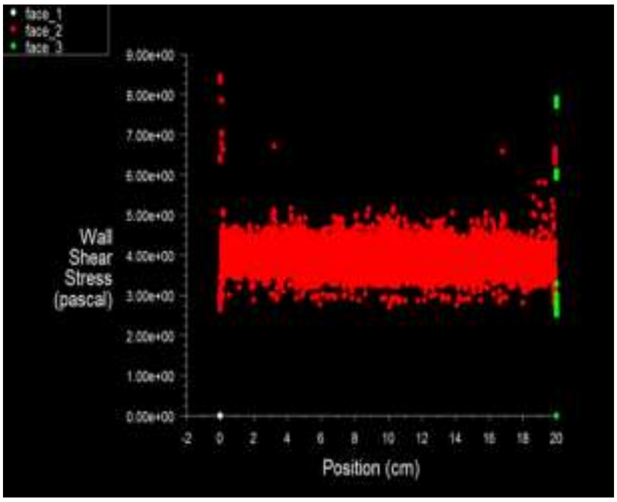

Fig.14. The real simulation wall shear stress of case A.

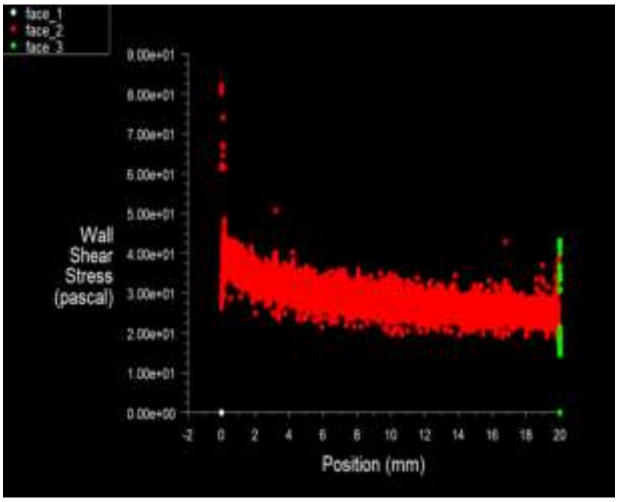

Fig.15. The real simulation wall shear stress of case B.

9. Conclusion

The aim of this study was investigating air flow through narrow tubes of different dimensions. Constant ratio of radius to length has been used. Cylindrical narrow tubes, represented as case A and case B were used. Case A was a tube of radius (2 cm) and length (20 cm) while case B was a tube of radius (2 mm) and length (20 mm). The initial conditions and boundary conditions for both cases were similar. Finite volume method has been used as a numerical technique by the aid of computer. Due to its widespread applications, air has been used as fluid material for the both cases. No slip at the tube wall has been used and viscous fluid flow condition was considered. Both cases performed under different Reynold's number (Re). The velocity profiles of fully developed flow were compared with the perfect parabolic velocity profile. Clear differences have been illustrated between the two cases A and B because of the narrowness of case B compared to case A. The maximum velocity was at the tube centerline and zero at the tubes wall because of no-slip condition has been used. Due to the differences in the diameters for both cases, the results showed apparent differences between the two cases for different physical quantities such as velocity magnitude, dynamic pressure, static pressure and wall shear stress as well. It could be conclude that manipulating the tube dimensions can results in useful fluid flow proprieties for different engineering applications.

Acknowledgements

The author gratefully acknowledges the Department of Physics, College of Education for Pure Science, University of Basrah.

References

[1] G. K. Batchelor, An Introduction to Fluid Dynamics, Cambridge Univ. Press, Cambridge, England, 1967.

[2] Y. A. Çengel, J. M. Cimbala, Fluid mechanics: fundamentals and applications, McGraw-Hill, (2018).

[3] Md. Alamin, Md. I. Hossain, Md. R. Hasan, Md. M. Rahman, K.C. Roy, A. Latif, Applying Laminar and Turbulent Flow and measuring velocity Profile Using MATLAB, IOSR Journal of Mathematics (IOSR-JM), 13, Issue 6 Ver. II,(2017), pp. 52-59.

[4] F. A. Morrison, An introduction to Fluid Mechanics, Cambridge University Press, (2013).

[5] B. Andersson, R. Andersson, L. Hakansson, M. Mortensen, R. Sudiyo, B. Van Wachem, Computational Fluid Dynamics for Engineers, Cambridge University Press, (2012).

[6] I. Okhotnikov, S. Noroozi, P.Sewell, P. Godfrey, Evaluation of steady flow torques and pressure losses in a rotary flow control valve by means of computational fluid dynamics, International Journal of Heat and Fluid Flow, 64, (2017), pp. 89-102.

[7] X. Zhang, J. Li, J. Zhu, L. Qiu, Computational fluid dynamics study on liquefied natural gas dispersion with phase change of water, International Journal of Heat and Mass Transfer, 91, (2015), pp. 347-354.

[8] B. L. da Silva, R. D. Luciano, J. Utzig, H. F. Meier, Analysis of flow behavior and fluid forces in large cylinder bundles by numerical simulations, International Journal of Heat and Fluid Flow, 75, (2019), pp.209-226.

[9] R. Morasata1, K. M. Güleren, Solution of the Two-Dimensional Steady State Heat Conduction using the Finite Volume Method, Ninth International Conference on Computational Fluid Dynamics (ICCFD9), Istanbul, Turkey, July 11-15, (2016), pp.1-7.

[10] A. Rouboa, E. Monteiro and R. D. Almeida, Finite volume method analysis of heat transfer problem using adapted strongly implicit procedure, Journal of Mechanical Science and Technology, 23 (2009), pp. 1553-1562.

[11] O. Zikanov, Essential Computational Fluid Dynamics, JOHN WILEY & SONS, INC. (2010).

[12] G. Kaoullas and G. C. Georgiou, Newtonian Poiseuille flows with slip and non-zero slip yield stress, Journal of Non-Newtonian Fluid Mechanics, Vol. 197, (2013), pp.24-30.

[13] S. Nallapua and G. Radhakrishnamachya, Jeffrey Fluid Flow Through A Narrow Tubes In The Presence of a Magnetic Field, Procedia Engineering Vol.127 (2015), pp.185 – 192.

[14] Ch. Stevenin, Y. B ereaux, J.-Y. Charmeau, Shaping Air Flow Characteristics of a High-Speed Rotary-Bell Sprayer for Automotive Painting Processes, Journal of Fluids Engineering, ASME, (2015), pp.111304-(1-8).

[15] Ling-Xiao Zhao, Chun-Lu Zhang, Liang-Liang Shao, Liang Yang, A Generalized Neural Network Model of Refrigerant Mass Flow Through Adiabatic Capillary Tubes and Short Tube Orifices, Journal of Fluids Engineering , ASME, (2007), pp. 1559 -1564.

NUMERICAL INVESTIGATION OF FLUID FLOW IN CAPILLARY TUBES

عامر حسين علي

قسم الفيزياء / كلية التربية للعلوم الصرفة / جامعة البصرة

الخلاصة

اهتمت هذه الدراسة على جريان المائع في الانابيب الضيقة. تم استخدام تقنية الحجم المتناهي كطريقة للتحليل العددي بمساعدة الكمبيوتر لتنفيذ الدراسة. تم استخدام انبوبين اسطوانيين ضيقيين اطلق عليهما الحالة A والحالة B . الحالة A هي انبوب بنصف قطر (2 cm) وطول (20 cm) بينما الحالة B والتي استخدمت لتفحص تاثير الانبوب الضيق على جريان المائع كانت بنصف قطر (2 mm) وطول (20 mm). الشروط الابتدائية والشروط الحدودية هي نفسها لكلتا الحالتين. بسبب الاستخدام الواسع للهواء تم استخدامه في كلتا الحالتين مائعا جاريا. استخدمت فرضية المائع الجاري اللزج وغير القابل للانزلاق عند الجدار. تم دراسة متغيرات مختلفة وتم مقارنتهم للحالتين مثل السرعة و الضغط الديناميكي و الضغط الستاتيكي واجهاد القص السطحي. تمتلك الحالة A عدد راينولد اكبر من الحالة B. شكل جبهة السرعة في منطقة النضوج تم مقارنتها مع جبهة سرعة ذات قطع مكافئ. وجد فرقا واضحا بين الحالتين بسبب الحالة B الاكثر ضيقا. السرعة القصوى تكون عند مركز الانبوب و صفر عند الجدار بسبب تطبيق حالة المائع غير القابل للانزلاق. بسبب الاختلاف في القطر للحالتين، اظهرت النتائج اختلاف واضح للسرعة والضغط الديناميكي و الضغط الستاتيكي واجهاد القص السطحي. نتائج هذه الدراسة تعتبر مهمة للتطبيقات التي تتضمن تدفق الهواء في انابيب ضيقة.

الكلمات الدليلية: طرق الحجم المتناهي ، جريان المائع، الانابيب الشعرية.