NUMERICAL SIMULATION OF SLIP BEHAVIORS AND FRICTION REDUCTION EFFECTS IN HYDROPHOBIC MICRO-CHANNEL IN LAMINAR FLOW CONDITIONS

by

Bekir DOGAN[a*], Mustafa OZBEY[b], Lutfu NAMLI [b], and Unsal AYBEK[a]

[a] Tokat Vocational School, Tokat Gaziosmanpasa University, Tokat, Turkey
[b] Faculty of Engineering, Ondokuz Mayis University, Samsun, Turkey

Original scientific paper
https://doi.org/10.2298/TSCI2304405D

In the study, a numerical simulation of the sliding properties of the rough and smooth surfaces with micro-structure was made. The simulation of shear flow in the micro-channel was performed with ANSYS FLUENT software. The 3-D and two-phase flow is simulated by choosing the volume of fluid model. In CFD analysis, water and air consist of two immiscible phases. In the calculations, if water is the first fluid and air is the second fluid, adjustments are made. At the beginning of the analysis, the channel was considered to be completely filled with air and the effect of gravity was ignored during the calculation. Water and air are considered Newtonian and incompressible fluids. In addition, laminar flow and steady-state calculations are made. It was found that the decrease in pressure drop increased with increasing distance between asperities (no-shear fraction). In the simulation results, approximately 14% of the velocity in the micro-channel axis was measured at the interface. The main purpose of this study is to evaluate the applicability of the volume of fluid model in a hydrophobic micro-channel flow designed in 3-D using ANSYS Fluent CFD software.

Key words: *ANSYS fluent, sliding speed, volume of fluid, interface, micro-channel*

Introduction

The usage areas of hydrophobic surfaces, which are the product of nanotechnology, are becoming more and more widespread. Self-cleaning surfaces, non-wetting fabrics, and non-stick kitchen utensils can be given as examples. In the industrial field, their use is increasing day by day to improve heat transfer, prevent corrosion, prevent icing, reduce friction resistance, generally save energy, and improve performance. The hydrophobic property increases with the surface roughness of the solid [1]. The contact angle measured on uneven surfaces does not exceed 120°. However, on solid surfaces with high roughness, the air between the surface of the solid and the liquid increases the hydrophobic property by reducing the interaction between the surfaces [2].

Optimization gas bubbles trapped within the nano-roughness of the hydrophobic surface significantly reduce the surface friction of the overlying liquid flows [3]. It is clear that there is a gas film consisting of bubbles at the solid-liquid interface in a flow occurring on a hydrophobic surface [4]. As seen in fig. 1, the gas film in the roughness transforms the solid-liquid interface into a gas-liquid interface. Thus, the contact area between the liquid and the solid wall and the frictional resistance of the liquid near the wall decrease [5].

* Corresponding author, e-mail: bekir.dogan@gop.edu.tr

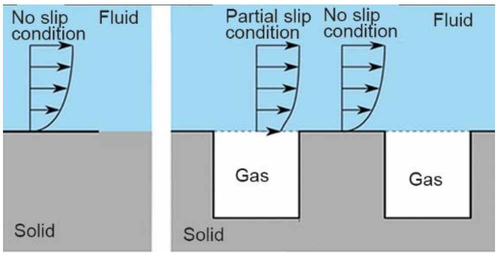


Figure 1. Schematic representation of fluid shear on a smooth and rough surface

Due to the viscous effect of the fluid, the fluid has a greater shear force near the wall [6]. The sliding speed reduces the shear force and the corresponding frictional resistance. Therefore, the presence of shear velocity contributes to the reduction of resistance [7]. It has been stated that the flow velocity on a hydrophobic surface is different from zero [8].

Various studies have been carried out by Watenabe *et al.* [9] on the shear boundary conditions on a solid surface. In their studies, they reported a 22% reduction in frictional resistance for laminar Newtonian flow in square and rectangular channels. Similarly, in the experimental study conducted by Hao *et al.* [10] they obtained a 10% to 30% reduction in frictional resistance for laminar flow in square and rectangular channels.

Design of the micro-channel and creation of the mathematical model of the flow field

Design of micro-channel

Flow and heat transfer in micro-channels has been a major area of interest. In the literature, channels with a hydraulic diameter greater than 3 mm are classified as conventional channels, channels between 3 mm and 200 μm are classified as mini-channels, and channels between 200 μm and 10 μm are classified as micro-channels [11]. Figure 2 shows the schematic diagram and some physical dimensions of the micro-channel.

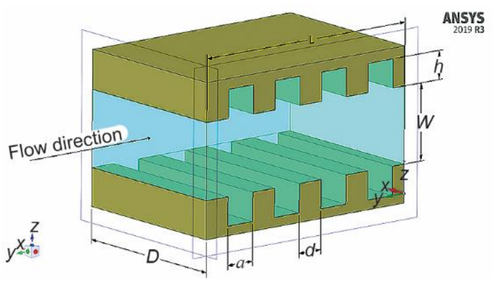


Figure 2. Design of the micro-channel

Microstructured channel: length L, channel height W, channel width D, inter-roughness width a, roughness thickness d, and roughness height h. For the smooth micro-channel a, h, and d were accepted as zero. Table 1 shows the geometric dimensions of the micro-channel structures used in this study.

Table 1. Geometric dimensions of micro-channels

Channel length, L [mm]	Channel width, D [μm]	Channel height, W [μm]	Roughness thickness, d [μm]	Interroughness width, a [μm]	Roughness height, h [μm]
24	180	102	0	0	0
24	180	102	28	32	30
24	180	102	28	62	30

Mesh design

A 3-D model of the fluid volume in the micro-channel was designed in the ANSYS DESIGN program. For the flow volume model created in the ANSYS DESIGN MODELLER section in fig. 3, a solution network was created in the ANSYS MESH section.

The network structure near the wall has a great influence on the flow field. Based on the properties of the flow volume, the mesh in the flow area is constructed using rectangular elements. In this context, first of all, mesh adherence test was performed and the number of meshes in which the solution was independent of mesh was determined [12].

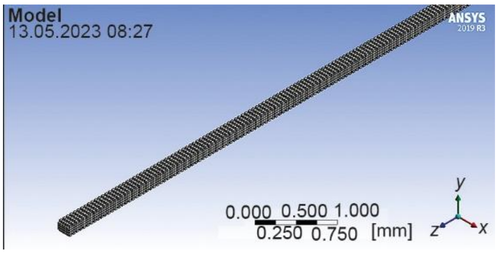


Figure 3. Micro-channel mesh image

At the end of the meshing process in ANSYS, the maximum *skewness* and the minimum *orthogonal quality* value are required to be in the acceptable range. As seen in tab. 2, it is desired that the maximum skewness value should not exceed 0.95 as much as possible, and the minimum orthogonal quality value should not fall below 0.1 [13].

Table 2. *Skewness orthogonal quality* network metrics spectrum

	Unacceptable	Bad	Acceptable	Good	Very good	Excellent
Maximum skewness	0.98-1.00	0.95-0.97	0.80-0.94	0.50-0.80	0.25-0.50	0-0.25
Minimum orthogonal quality	0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

In tab. 3, the number of elements used, the maximum *skewness* and minimum *orthogonal quality* values of this element structure are given. The values in our mesh structure show that the mesh is also of excellent quality.

Table 3. Mesh structure quality criteria

Number of nodes	Number of elements	Maximum skewness	Minimum orthogonal quality
19224	11335	1.3073e-10	1

Numerical method

In the study, FLUENT calculations were made by using ANSYS 2019 R3 CFD software. In ANSYS FLUENT, the volume of fluid (VoF) two-phase flow model is used to monitor the gas-liquid interface [14]. Flow calculation information is given in tab. 4.

At the beginning of the analysis, the channel was considered to be completely filled with air and the effect of gravity was ignored during the calculation. The performed CFD analysis consists of two phases that do not interfere with each other. Adjustments are made in the CFD flow as the water primary phase and air as the secondary phase. Water and air are considered Newtonian and incompressible fluids. In addition, laminar flow and steady-state calculations are made.

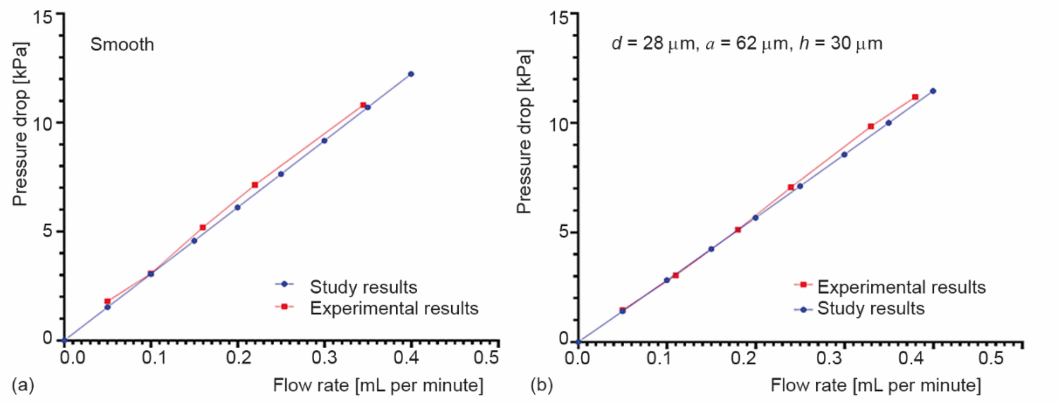
The main purpose of this study is to evaluate the applicability of the VOF model in a hydrophobic micro-channel flow designed in 3-D using ANSYS FLUENT CFD software.

Table 4. Simulation setting

Multiphase flow model	The VoF
Primary phases	Water
Water density	998.2 kg/m^3
Water viscosity	0.001003 Pa s
Secondary phase	Air
Air density	1.225 kg/m^3
Air viscosity	1.7894×10^{-5} Pa·s
Surface tension	0.073 N/m
Flow type	Laminar flow
Calculation type	Steady model
Input setting	Velocity inlet
Output setting	Pressure outlet

Results and discussion

Hao *et al.* [10] experimentally investigated pressure drops in hydrophobic micro-channels under laminar flow conditions. In this article, the pressure drop values obtained using ANSYS FLUENT CFD software are compared with the experimental results. As seen in fig. 4, Hao *et al.* [10] experimental and numerical results of the presented study were found to be compatible and consistent. Looking at these results, it is seen that the work done with the mesh quality and VOF model gives results close to the experimental data.



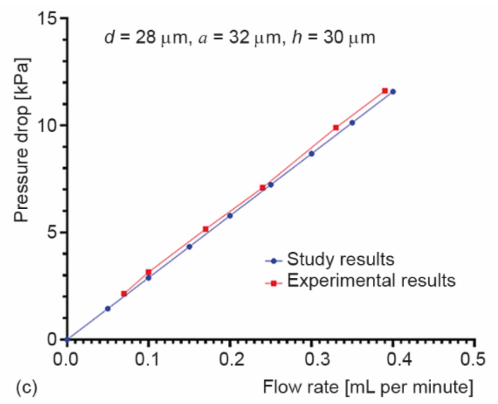


Figure 4. Validation charts

Calculations were made with eight different flow rates for the same geometry. The mesh structure and analysis steps on the calculated geometry are exactly the same. The pressure drop values obtained as a result of the analysis are presented in tab. 5.

In fig. 5, it was determined that the pressure drop increases linearly as the flow rate increases. Compared to the smooth hydrophobic micro-channel, a higher friction reduction was observed on the surface of the other two rough micro-channels.

It has been stated that creating micro-sized rough structures on the surface is an effective method to maintain the air-water inter-

Table 5. Pressure drop analysis results at different flow rates

Flow rate [mL per minute]	Pressure drop [kPa]		
	smooth	d = 28 μm, a = 32, μm h = 30 μm	d = 28 μm, a = 62, μm h = 30 μm
0.05	1.525	1.444	1.404
0.1	3.052	2.889	2.820
0.15	4.580	4.335	4.246
0.2	6.108	5.782	5.679
0.25	7.638	7.230	7.117
0.3	9.169	8.679	8.561
0.35	10.701	10.129	10.009
0.4	12.233	11.580	11.462

face [15, 16]. Figure 6 shows the air plastron state on micro-channel surfaces with roughness in both micro dimensions. As can be clearly seen in fig. 6 of the micro-channel with dimensions, d = 28 μm, a = 32 μm, and h = 30 μm, it is seen that air remains between the asperities during the flow [10]. The air plastron completely collapsed in the micro-channel with dimensions, d = 28 μm, a = 62 μm, and h = 30 μm. With the increase in the roughness width (a), it was observed that all the air in the pore was discharged during the flow, fig. 6.

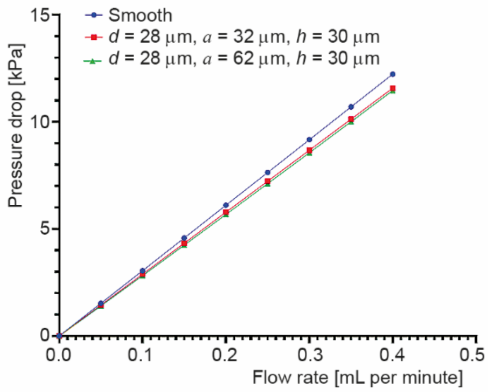


Figure 5. Pressure drops on surfaces at different flow rates

For velocity measurements, plane and line definitions were made in the ANSYS FLUENT CFD post section to pass through the channel axis. The view of the velocity profile on the plane is shown in fig. 7. Slip rates were obtained on rough surfaces with microstructure. The velocity at the air-water interface is not zero. The sliding velocity at the interface was measured at approximately 14% of the velocity in the micro-channel axis.

Conclusions

Significant shear effects and pressure drops on the smooth and regular rough surfaces of the hydrophobic micro-channel designed in 3-D with ANSYS FLUENT CFD software under laminar flow conditions were investigated. In this study, the applicability of the VOF model was tested. Compared with the literature results, the results were found to be approximately the experimental results. Pressure drop reductions were observed in the simulation results. Shear velocity values on the wall with regular roughness confirmed the pressure drop reductions. The sliding velocity at the interface was measured at approximately 14% of the velocity in the micro-channel axis. It has been shown that the decrease in pressure drop increases with increasing distance between asperities (no-shear fraction). The air-water interface

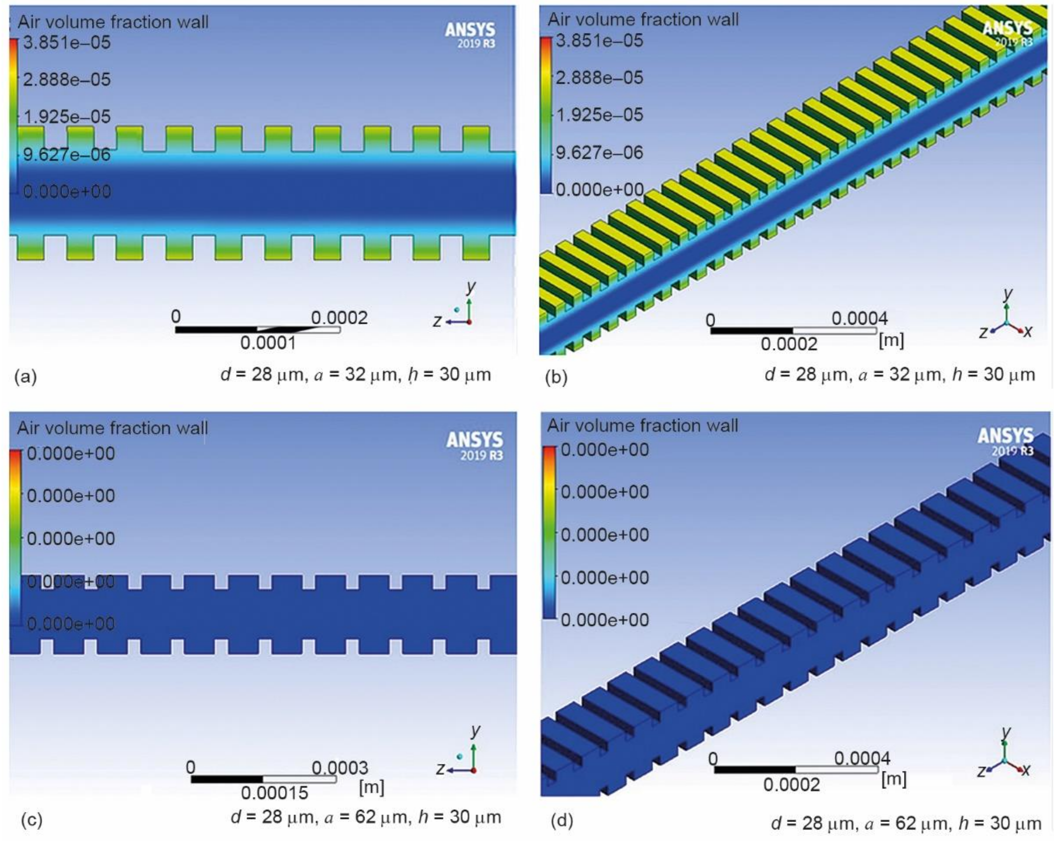


Figure 6. Air volume fraction images

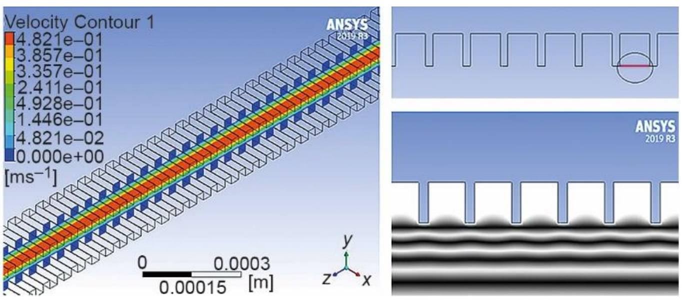


Figure 7. Speed contours on plane and line

is clearly visualized in the simulation of the micro-channel with dimensions d = 28 μm, a = 32 μm, and h = 30 μm.

In summary, the micro-channel surface with regular roughness has a relatively significant friction reduction effect under laminar flow conditions. It is necessary to investigate and develop the friction reduction effect of surfaces in turbulent flow conditions. It is of great importance to develop a model suitable for different flow regimes. Lowering the drag resistance is of great importance in areas where the surface is in contact with liquids such as pipelines, aircraft wings, ships, submarines, and torpedoes.

References

[1] Choi, C.-H., *et al*., Effective Slip and Friction Reduction in Nanograted Superhydrophobic Microchannels, *Physics of Fluids, 18* (2006), 8, 087105
[2] Kartal, F. E., Synthesis and Characterızatıon of Nanocomposite Superhydrofobic Surfaces, M.Sc. thesis, Gazi University, Ankara, Turkey, 2009
[3] Picella, F., *et al*., Laminar-Turbulent Transition in Channel Flow with Superhydrophobic Surfaces Modelled as a Partial Slip Wall, *J. Fluid Mech., 881* (2019), Oct., pp. 462-497
[4] Picella, F., *et al*., On the Influence of the Modelling of Superhydrophobic Surfaces on Laminar-Turbulent Transition, *J. Fluid Mech., 901* (2020), 901, A15-36
[5] Wang, J., *et al*., Slip Theory-Based Numerical Investigation of the Fluid Transport Behavior on a Surface with a Biomimetic Microstructure, *Engineering Applications of Computational Fluid Mechanics, 13* (2019), 1, pp. 609-622
[6] Pehlivan, M., Ozbey, M., Experimental Investigation of the Effect of Hydrophobic Surfaces on Friction Losses in Turbulent Pipe Flows, *Duzce University Journal of Science and Technology, 10* (2022), 3, pp. 1383-1397
[7] Kibar, A. Experimental Investigation of Behaviour of Water Jet Flow onto Hydrophobic Surfaces, M.Sc. thesis, Kocaeli University, Kocaeli, Turkey, 2008
[8] Wang, C., *et al*., Spray-Coated Superhydrophobic Surfaces with Wear-Resistance, Drag-Reduction and Anti-Corrosion Properties. Colloids and Surfaces A: Physicochem, *Eng. Aspects, 514* (2017), Feb., pp. 236-242
[9] Watanabe, K., *et al*., Drag Reduction in Flow through Square and Rectangular Ducts with Highly Water-Repellent Walls., *Transactions of the Japan Society of Mechanical Engineers, 62* (1996), Feb., pp. 3330-3334
[10] Hao, P., *et al*., Laminar Drag Reduction in Hydrophobic Microchannels, *Chem. Eng. Technol., 32* (2009), 6, pp. 912-918
[11] Kandlikar, S G., Grande, W. J., Evolution of Microchannel Flow Passages-Thermohydraulic Performance and Fabrication Technology, *Heat Transfer Eng., 24* (2003), 1, pp. 3-17
[12] Paz, C., *et al*., Analysis of the Volume of Fluid (VOF) Method for the Simulation of the Mucus Clearance Process with CFD, *Computer Methods in Biomechanics and Biomedical Engineering, 22* (2019), 5, pp. 547-566
[13] Fatchurrohman N., Chia S.T., Performance of Hybrid Nano-Micro Reinforced Mg Metal Matrix Composites Brake Calliper: Simulation Approach, *IOP Conf. Ser.: Mater. Sci. Eng., 257* (2017), Aug., 012060
[14] Zhu, J., *et al*., Numerical Simulation of Slip Characteristics of Superhydrophobic Surface, *Proceedings,* ACM ISBDAI Conference (ISBDAI'20), ACM, Johannesburg, South Africa, 2020, pp. 366-370
[15] Bhushan, B., *et al*., Fabrication and Characterization of the Hierarchical Structure for Superhydrophobicity and Self-Cleaning, *Ultramicroscopy, 109* (2009), 8, pp. 1029-1034
[16] Zhang, J., *et al*., Drag Reductions and the Air-Water Interface Stability of Superhydrophobic Surfaces in Rectangular Channel Flow, *Physical Review E, 94* (2016), 5, 053117

Paper submitted: February 15, 2023
Paper revised: April 28, 2023
Paper accepted: June 10, 2023