American Journal of Science and Technology
2015; 2(3): 106-110
Published online April 30, 2015 (http://www.aascit.org/journal/ajst)
ISSN: 2375-3846

American Association for Science and Technology

Drag Reduction by Fluid Slip

K. Watanabe

Science and Engineering Research Institute of Kokushikan University, 4-28-1 Setagaya, Setagaya-ku, Tokyo, JAPAN

Email address

keizo@tmu.ac.jp

Citation

K. Watanabe. Drag Reduction by Fluid Slip. *American Journal of Science and Technology*. Vol. 2, No. 3, 2015, pp. 106-110.

Keywords

Fluid Mechanics,
Newtonian Fluids,
Fluid Slip,
Laminar Drag Reduction,
Circular Pipe Flow,
Friction Factor,
Flow past a Sphere,
Drag Coefficient

Received: March 29, 2015
Revised: April 13, 2015
Accepted: April 14, 2015

Abstract

Drag reduction technique is receiving a lot of attention from view point of energy saving in fluid engineering. It can be considered that the target is a turbulence modification for turbulent flow in order to reduce the pressure loss or drag. However, it is no effect for laminar flow range because the turbulence does not exist in the flow. If fluid slip occurs at the wall surface, we can obtain the drag reduction even if it is the laminar flow range. In this paper, some experimental results of laminar drag reduction are described and discussed for its application.

1. Introduction

In recent years, the drag reduction phenomenon attracts attention in the field of various kinds of industry from view point of the energy saving. In addition, the research to be related to it is hydrodynamically receiving a lot of attention in terms of the flow control. The technique to get the drag reduction can be classified into two categories, namely, the passive and the active control. In the former case, we control the flow by changing fluid characteristics or surface structure, therefore, there is no necessary to supply energy from the outside of flow system. It is a popular technique in the drag reduction. Because turbulent flow is a flow regime by characterized chaotic property changes, the target becomes the turbulence modification in order to obtain the drag reduction in the flow region. Method for adding drag reducing additives namely high molecular weight polymer or using riblet is a major example in the passive control. They are often called the turbulent drag reduction. However, it is obvious that these techniques are not applicable for laminar flow because the flow is non turbulent streamline flow in parallel layers. Thus, the study has been accomplished to turbulent flow in the past. The phenomenon of resistance to motion through a fluid is occurred due to fluid-wall interaction, and a real fluid does not usually slip on the wall in contact with it. It is known as no slip boundary condition and the pressure drop or the friction drag has been analyzed using the no slip boundary condition. If fluid slip occurs at the solid boundary, the drag reduction occurs even in laminar flow range. This paper describes the typical examples of laminar drag reduction (1) for a Newtonian liquid flow.

2. Boundary Condition

2.1. Fluid Slip

As mentioned above, many experimental results of Newtonian fluids flow agree well with the analytical result using no fluid slip boundary condition, and it is almost accepted that fluid slip does not occur on the wall in the continuum hydrodynamics now. However, it has been reported that fluid slip (2) ~ (5) occurs in a liquid flow of a micro channel with certain type of a wall surface although the phenomenon did not receive much attention for engineering application because the order of the slip velocity is micro scale. The contact angle of the channel wall is larger than that of usual wall. Those walls are often called a hydrophobic wall or a highly water-repellent wall. The fluid slip phenomenon of liquid flow occurs in not only a micro channel but also a pipe [6] of millimeter order in the flow system size. The deference for the order of slip velocity depends on the hydrophobic wall structure.

Although the contact angle is defined as the physical constant representing a physical property of the hydrophobic wall, it is insufficiently for the physical constant to grasp the fluid slip phenomenon. Identification of the physical constant is necessary with the clarification of the mechanism that the fluid slip occurs at the wall.

2.2. Slip Velocity

It is necessary to use the slip boundary condition in order to analyze the flow phenomenon if fluid slip occurs at the wall.

Navier [7] gives the hypothesis on fluid slip. It is expressed as following equation,

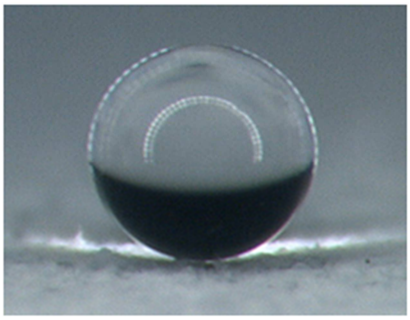
$$u_s = \frac{\mu}{\beta}\left|\frac{\partial u}{\partial y}\right| \tag{1}$$

Where, u_s is the slip velocity at the position of a prescribed distance from the wall y. μ and β are viscosity and sliding constant, respectively. In no slip case, it is given by the substituting $\beta \to \infty$ for Eq. (1). Eq. (1) gives the relationship between the slip velocity and the wall shear stress τ_w at the wall surface. Thus we can drive the exact solution of Navier-Stokes equation for a pipe or a channel flow with fluid slip using Eq. (1).

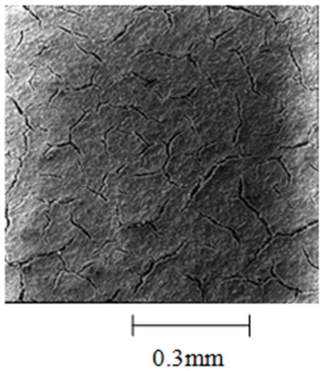
Eq.(1) expresses that the slip velocity is proportional to the wall shear stress at the wall if β is given as the constant value. In other word, we can decide the value of the sliding constant from the measurement value of the slip velocity and the wall shear stress. It is clarified by the experiment that τ_w and u_s satisfies liner relation [6] for Newtonian fluid flow. Other hand, it is nonlinearity in case of a high-molecular-polymer solutions flow in a circular pipe [8].

The sliding constant of the order is $O(1) \sim O(10)$ for water flow in a pipe. But, the difficulty remains in the determination of β quantitatively because there are very few experimental data for the slip velocity. The existence of nano-bubbles at hydrophobic wall has been identified as the cause that slip occurs for a long time. In particular, even though such bubbles have been observed, the effect has no impact in case of relatively large flow system size. It is necessary to accumulate the experimental data for the slip velocity.

Figs. 1(a) and 1(b) show the configuration of a droplet at the highly water-repellent wall that was used for the study [6] on large system size and the micrograph of the surface, respectively. It is seen that the wall obtaining grater slip velocity has many cracks as microscopic grooves at the surface. It is seen also that the slip velocity does not occur at the highly water-repellent wall without the fine grooves. Thus, a surface that forms between a liquid and a gas plays an important role for the possibility of slip. By consideration the configuration of solid boundary, Lauga and Stone [9] propose a solid boundary model of uniform slip and no slip region along with wall for slip flow. Fujita and Watanabe [10] obtain a numerical simulation result of flow past a sphere with hydrophobic wall using a liquid -gas solid boundary model.



(a) Configuration of a droplet



(b) Macrograph of wall surface

Fig. 1. Highly water-repellent wall

3. Experimental Results

3.1. Internal Flow

A pipe or duct is the main constituting elements of fluid transporting apparatus. Thus, the application of the laminar drag reduction phenomenon to these pipelines has a high industrial utility value if a relatively large slip velocity is generated at these walls.

Fig. 2 shows an example of the experimental results [6] for the friction factor of a circular pipe with a highly water repellent-wall shown in Fig. 1(b). In Fig. 2, the friction factor is given as $\lambda = \Delta p \big/ (l/2a)\left(\rho U_0{}^2/2\right)$. Where, $\Delta p, U_0$ and ρ are the pressure loss, mean velocity and density, respectively. a and l are the side wall length and the measurement section for the pressure loss, respectively. The solid lines in

Fig. 2 are an exact solution of laminar flow and Blasius formula for turbulent flow obtained using no slip boundary condition, respectively. Dotted line is the analytical result obtained using Eq. (1) under the slip boundary condition at the wall. It is seen that the drag reduction occurs in laminar flow range.

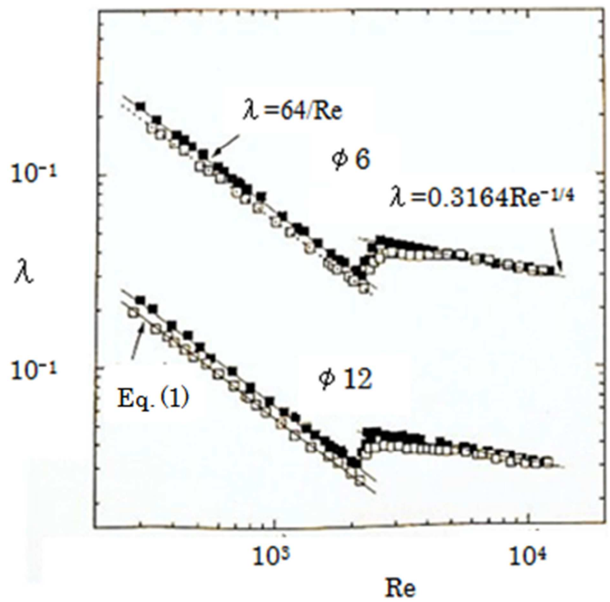
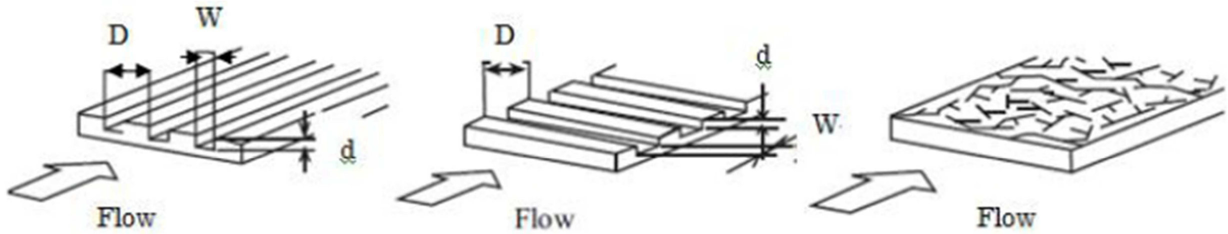


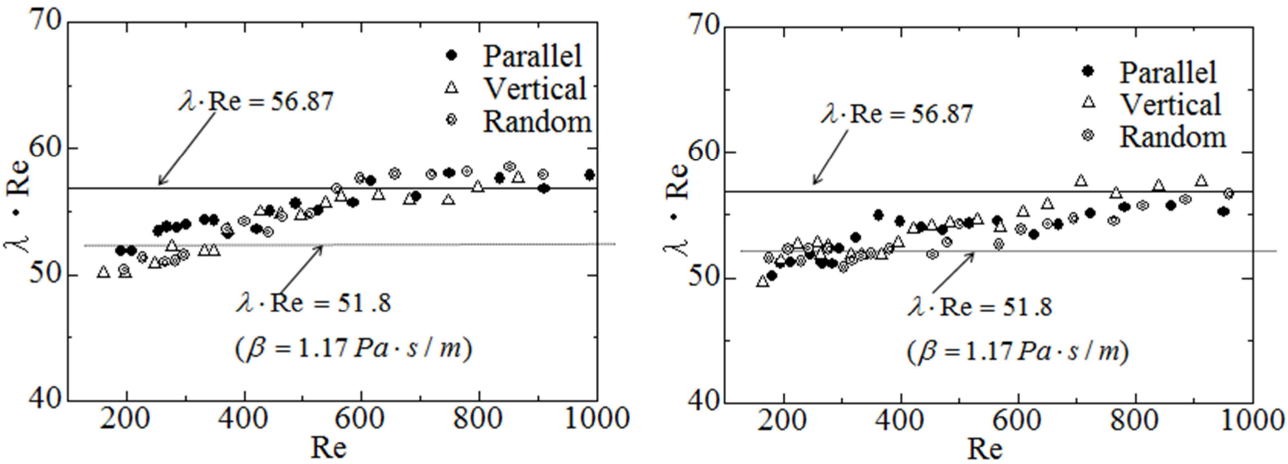
***Fig. 2.** Friction factor of a circular pipe*

Referring to Fig. 1(b), we can make the drag reducing wall that gas-liquid interface retention is possible. Accordingly, Watanabe et al. [10] provided a highly water repellent wall with a fractal structure. Fig. 3 shows the sketch drawing of the wall. They are made of silicon and these grooves are subjected to wet etching after patterning by ultraviolet rays irradiation. The depth d, is 20μm. There are two kinds of the wall with 5μm and 10μm in the width for each wall. It is 15% in an area ratio of the groove referring to from the result in Fig. 1(b). Experiments were performed about the measurement of the pressure loss of a duct with the test wall as shown in Fig. 4. The experimental results are shown in Fig. 4. The solid line in Fig. 4 is an exact solution of a square duct obtained using no slip boundary condition, $\lambda \cdot \mathrm{Re} = 56.87$. The dotted line is the analytical result that fluid slip occurs at one side of the wall by using Eq. (1). The wall exhibits drag reducing effect under conditions both $5\mu m$ and $10\mu m$ the width. It is 13% in the maximum drag reduction ratio in the range of $150 \le \mathrm{Re} \le 770$. However, the experimental data increase with the increase of Reynolds number and they coincide with the result for no-slip boundary condition at certain Reynolds number range. The trend of the experimental data is caused by disappearance of the gas-liquid interface at the wall. That means interfacial disappearance at the wall with an increase in Reynolds number. It is a future research theme to make the highly water repellent wall where stable interfacial maintenance is possible.



(a) Parallel grooves (b) Vertical grooves (c) Random grooves

***Fig. 3.** Sketch drawing of test wall*

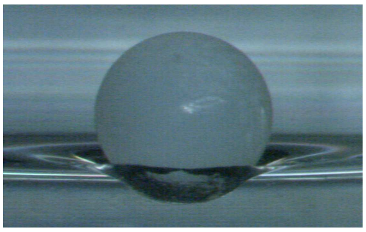


(a) 10μm width groove (b) 5μm width groove

***Fig. 4.** Friction factor of test wall*

3.2. External Flow

Flow around a sphere is a typical example of the external flow, and many studies have been conducted into the flow behavior. In addition, the flow pattern or the relationship between the drag coefficient and Reynolds number under no slip boundary condition is experimentally clarified in wide Reynolds number range. By this constitution, a detailed comparison between and no slip flow and slip flow is enabled. The sphere with highly water-repellent wall is covered on a stable gas film when it is immersed in quiescent water as shown in Fig. 5(a). According to the requirement we can make a sphere (11) where the sedimentation does not occur. Fig. 5(b) shows the sphere which can prevent the settling in a vessel filled with water.

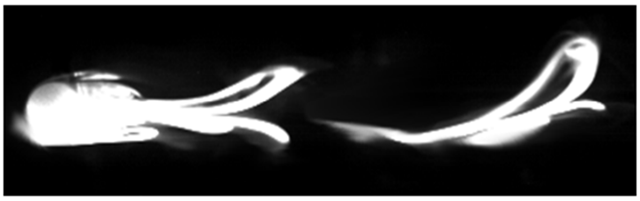


(a) 50mm, ρ=1130kg/m^3

(b) d=2.39mm, ρ=1130kg/m^3

Fig. 5. *Highly water-repellent wall sphere in water that is at rest*

On the other hand, a marked difference occurs between flow around a sphere with highly water-repellent wall and a smooth wall sphere when it remains stationary in flow-field. Fig.6 shows the experimental visualization results for the flow where it is $\mathrm{Re} = 320$. As it is clear if you compare these two photographs, the generation of the vortex flow from the periphery of the sphere is suppressed thereby. As a matter of course, the change of the wake leads to the drag reduction in the flow range. Experimental results (12) of the drag coefficient of a sphere are plotted in Fig. 7. In Fig. 7, the solid line (13) is a curve that was summarized the experimental results of the past for a smooth wall sphere. In addition, the numerical simulation results (14) calculated using gas-liquid interface model at solid surface are shown in the figure. As shown in Fig.8, the drag reduction occurs in the range of $\mathrm{Re} \le 3000$. It is $\mathrm{DR} = 28.5\%$ in the drag reduction ration for $\mathrm{Re} = 7.2$. The analytical result which influence of fluid slip is taken into consideration of the boundary condition, agrees well with the experimental result.



(a) Smooth sphere

(b) Highly water-repellent wall sphere

Fig. 6. *Vortex loop and vortex ring behind sphere at Re= 320.*

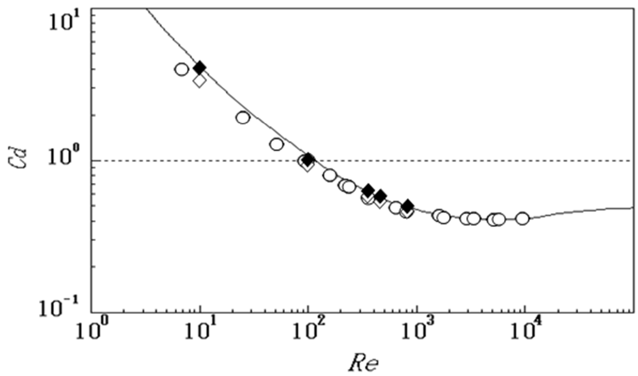


Fig. 7. *Drag coefficient versus Reynolds number; ○, experimental data of a highly water-repellent wall sphere (12), ◇, numerical result of a highly water-repellent wall sphere, ◆, numerical result of a smooth surface sphere (14); —, Lapple and Shepherd (13)*

4. Summary

Drag reduction phenomenon of a highly water-repellent wall were presented referring the experimental results for a pipe flow and a flow around a sphere. In the analytical consideration, an exact solution was obtained for some laminar flow by Navier's hypothesis. However the difficulty remains in the numerical prediction for the sliding constant in the equation. It was experimentally clarified that the laminar drag reduction was caused by the fluid slip resulting from slip velocity generated by a gas-liquid interface at the wall with fractal structure. Thus, it is necessary to make the drag reducing wall where the gas-liquid interface is maintained in a wide Reynolds number range in order to apply it to actual pipeline system in future.

References

[1] Watanabe, K., (2015), Laminar Drag Reduction, Bentham Sci. Pub. Ltd, 1-105.

[2] Schnell, E., (1956), Appl. Phys., 27, 1149-1152.

[3] Churaev, N. V., Sobolev, V. D. and Somov, A. N., (1984) ,J.Colloid. Interface Sci. 97, 574-581.

[4] Tretheway, D. C., and Meinhert, C. D., (2002) , Phys. Fluids 14, L9.

[5] Luck, S., Mutharasan, R., and Apelian, D. (1987), Ind. Eng. Chem. Res., 26 (8), 1609-1616.

[6] Watanabe, K., Yanuar and Udagawa, Y., (1999) J. Fluid Mech., 381, 225-238.

[7] Navier, C.L.M.H., (1823), Memoires de l'Academie Royale des Sciences de l'Institut de France, 6, 389-440.

[8] Watanabe, K., and Udagawa, H., (2001), AICh E.J., 26, pp.729^739.

[9] Luaga, E., and Stone, H.A.(2003), J.Fluid Mech., 489, 55-77

[10] Watanabe, K., et al.,(2005)，Trans. of JSME (Se. B),71-712, 2843-2847.

[11] Watanabe, K., and Ogata, S., (2002), Patent in Japan, 2002-115591.

[12] Fujita、T., and Watanabe, K., (2004), Proc. of 2004 ASME Heat Transfer/Fluids Engineering Summer Conference, HT-FED2004-56046, 1-6.

[13] Lapple, C. E. , and Shepherd, C. B., (1940), *Indust. Eng. Chem.*, 32, Vol 5, 605-617.

[14] Fujita, T., and Watanabe, K., (2004), ASME IMEC&E '04, IMECE2004-62223, 1-6.