The Effect of Surface Gas/Liquid Entrapment on Drag Reduction

Haosen Tan
Southern Methodist University, tanhaosen@gmail.com

Follow this and additional works at: https://scholar.smu.edu/engineering_mechanical_etds

Part of the Fluid Dynamics Commons, and the Manufacturing Commons

Recommended Citation
Tan, Haosen, "The Effect of Surface Gas/Liquid Entrapment on Drag Reduction" (2022). Mechanical Engineering Research Theses and Dissertations. 43.
https://scholar.smu.edu/engineering_mechanical_etds/43

This Dissertation is brought to you for free and open access by the Mechanical Engineering at SMU Scholar. It has been accepted for inclusion in Mechanical Engineering Research Theses and Dissertations by an authorized administrator of SMU Scholar. For more information, please visit http://digitalrepository.smu.edu.
THE EFFECT OF SURFACE GAS/LIQUID ENTRAPMENT ON DRAG REDUCTION

Approved by:

Dr. David A. Willis
Associate Professor of Mechanical Engineering
Dissertation Committee Chairperson

Dr. Paul S. Krueger
Professor of Mechanical Engineering

Dr. MinJun Kim
Professor of Mechanical Engineering

Dr. Xu Nie
Associate Professor of Mechanical Engineering

Dr. Gary A. Evans
Professor of Electrical and Computer Engineering
THE EFFECT OF SURFACE GAS/LIQUID ENTRAPMENT ON DRAG REDUCTION

A Dissertation Presented to the Graduate Faculty of the
Bobby B. Lyle School of Engineering
Southern Methodist University

in
Partial Fulfillment of the Requirements
for the degree of
Doctor of Philosophy

with a
Major in Mechanical Engineering

by

Haosen Tan

B.S., Optical Information Science and Technology,
University of Science and Technology of China, 2016

May 14, 2022
ACKNOWLEDGMENTS

This thesis is dedicated to the individuals who have made direct or indirect contributions during the course of my research in the past 5 years. Without their insights and kind help, the completion of this work would certainly have been unimaginable. Among those many who deserve my sincerest gratitude and appreciation, I would like to first thank Professor David A. Willis, who is my Ph.D. advisor as much as he is a good friend to me, for his continuous support and advice from which I will surely take benefit for the rest of my career. My special thanks to Professors Paul S. Krueger, MinJun Kim, Xu Nie, and Gary A. Evans who have served as my graduate committee members.

I should also thank Bin Xia and Jie Sheng, my fellow ME students, for their friendship and help in many aspects, and Patrick Micheal Hillyard, Ian Cohen, and Alexander Brandt, my fellow lab mates for always being there while extra hands are needed.

I should also thank Kenny Sangston and Necdet Yildirimer for their help in making the parts and for laboratory maintenance.

I should thank Mr. James Dees for his assistance with the paperwork for graduation and formatting this dissertation.

This research is based on work supported by National Science Foundation (NSF) under grant no. 151070. The Moody School of Graduate and Advanced Studies is gratefully acknowledged for a fellowship that supported my final year of study.
Economic globalization has made today’s world more closely linked, where transportation (e.g. shipping, pipeline) has played an important role. Approximately over 60% (in air), 80% (underwater), and 100% (in pipeline) propulsive power is used to overcome surface drag, so reducing surface drag will have a substantial impact economically and environmentally.

To develop a surface that has lower drag, the intuitive idea is to convert no-slip boundary conditions to slip or partial slip ones. To achieve that, the most promising approach is to replace the solid-liquid boundaries completely or partially with gas-liquid or liquid-liquid boundaries, because the shear stress is much smaller at the gas-liquid and liquid-liquid interfaces compared to that at solid-liquid interfaces.

In this work, a perforated surface that can effectively trap gas (air) or liquid (water) near the surface was designed, fabricated, and characterized with a customized test configuration. An analytical model was developed to extract the representative quantities that reflect the overall slip effects, which was used to analyze the experimental data. Within the mass flow rate range of $1.382 \text{ g/s} \sim 2.764 \text{ g/s}$, the drag reduction observed in water-filled cases was up to 66%, and such a large drag reduction was due to bypass flow, which was confirmed by experiments and simulations. As for the air-filled cases, the air was effectively trapped in the holes, and the air-water interface was able to resist the water pressure and viscous shear forces even at the highest mass flow rate, 2.764 g/s. The drag reduction range was from 22% to 34% for air-filled cases, which corresponds to the effective slip length from $90 \mu m \sim 180 \mu m$. Such a large drag reduction and effective slip length were achieved with the large
feature size (up to 1 mm) and without chemically modifying the surface, which is favored by many application scenarios. In addition, numerical simulations were utilized to have more insight into the flow behaviors, such as the effect of the bypass flow on flow resistance, the effect of the air-water interface shape on the effective slip length, and the effect of the shape and layout of the pattern on the effective slip length.
TABLE OF CONTENTS

LIST OF FIGURES ................................................................. x
LIST OF TABLES ................................................................. xiv

CHAPTER

1. INTRODUCTION AND BACKGROUND ........................................... 1
   1.1. Motivation ................................................................. 1
   1.2. Research Background .................................................... 2
       1.2.1. Surfaces with drag reduction effect .............................. 2
       1.2.2. Metric of surfaces with drag reduction effect ............... 3
       1.2.3. Analytical model with Navier-slip conditions (Navier model) 6
       1.2.4. Analytical formulas for effective slip length ............... 7
   1.3. Overview ................................................................. 9

2. ANALYTICAL MODEL FOR SLIP DUCT FLOW .................................. 10
   2.1. Introduction ............................................................. 10
   2.2. Methods ................................................................. 11
       2.2.1. General boundary condition ...................................... 11
       2.2.2. Fully-developed laminar flow in a rectangular duct ........ 12
       2.2.3. Fully-developed laminar flow between two parallel plates .. 17
   2.3. Results ................................................................. 18
   2.4. Conclusion ............................................................. 23

3. EXPERIMENTAL AND NUMERICAL METHODS .................................... 24
   3.1. Surface design and fabrication ....................................... 24
       3.1.1. The design of a surface that can entrap gas or liquid ....... 24
       3.1.2. Generic pattern fabrication via laser micromachining system ... 27
3.2. Effective slip length estimation and measurement .......................... 30
  3.2.1. Analytical models for a fully developed laminar flow ................. 30
  3.2.2. Computational fluid dynamics (CFD) models .......................... 33
  3.2.3. Effective slip length estimation with computational fluid dynamics model ................................................................. 33
  3.2.4. Methods for the effective slip length calculation ......................... 34
  3.2.5. Design and construction of a system for measuring pressure loss (preliminary) ............................................................. 35
  3.2.6. Design and construction of a system for measuring pressure loss (final) 37

4. RESULTS AND DISCUSSION ......................................................... 40
  4.1. Challenges and significance .................................................. 40
  4.2. Characteristic parameters ..................................................... 44
  4.3. Experimental procedures ...................................................... 46
  4.4. Uncertainty analysis ............................................................ 51
  4.5. Experimental results ............................................................ 53
    4.5.1. Overview ................................................................. 53
    4.5.2. Lubrication cavity filled with water .................................. 54
    4.5.3. Lubrication cavity filled with air .................................... 57
  4.6. Simulation conditions .......................................................... 64
    4.6.1. The full-scale model .................................................... 65
    4.6.2. The unit cell model ..................................................... 66
  4.7. Simulation results ............................................................. 67
    4.7.1. The effects of bypass flow on drag reduction ......................... 67
    4.7.2. The effects of the air-water interface shape on the effective slip length 70
    4.7.3. The effects of the surface design on the effective slip length .... 72

5. CONCLUSIONS AND FUTURE WORK ............................................. 75
## LIST OF FIGURES

<table>
<thead>
<tr>
<th>Figure</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.1. Liquid flow over a solid surface with a slip quantified as slip length $b$. (a) A liquid flow over a hydrophobic solid surface with a negligibly small (tens of nanometers) slip length. (b) When a composite interface with solid and gas is formed on a structured surface, a liquid may flow with a relatively large effective slip length [24].</td>
<td>4</td>
</tr>
<tr>
<td>1.2. Experimental techniques used for the slip measurements and surface structures (both regular and random structures) probed by each [24].</td>
<td>5</td>
</tr>
<tr>
<td>1.3. The simple surface patterns and their corresponding formulas for theoretical predictions: (a) Grates parallel to the liquid flow, Equation (1.4a). (b) Grates transverse Equation (1.4b). (c) Posts, Equation (1.4c). (d) Holes, Equation (1.4d). [24]</td>
<td>8</td>
</tr>
<tr>
<td>2.1. Schematic of the model: A rectangular duct with a height of $2h$ and a width of $2w$, with fully-developed, incompressible, parallel, steady, laminar flow along the $x$-direction. Equation (2.1) is applied to the surfaces at $y = \pm h$.</td>
<td>13</td>
</tr>
<tr>
<td>2.2. Schematic of the model: Two parallel plates with a spacing of $2h$, with fully-developed, incompressible, steady, laminar flow along the $x$-direction. Equation (2.1) is applied to the walls at $y = \pm h$.</td>
<td>17</td>
</tr>
<tr>
<td>2.3. Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 1$. (a) no-slip, Navier-slip ($b = 50 , \mu$m), and fixed velocity ($u = 1.0 , \text{m/s}$); (b) no-shear.</td>
<td>19</td>
</tr>
<tr>
<td>2.4. Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 2$. (a) no-slip, Navier-slip ($b = 50 , \mu$m), and fixed velocity ($u = 1.0 , \text{m/s}$); (b) no-shear.</td>
<td>20</td>
</tr>
</tbody>
</table>
2.5. Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 5$. (a) no-slip, Navier-slip ($b = 50 \, \mu m$), and fixed velocity ($u = 1.0 \, m/s$); (b) no-shear. 20

2.6. Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 10$. (a) no-slip, Navier-slip ($b = 50 \, \mu m$), and fixed velocity ($u = 1.0 \, m/s$); (b) no-shear 21

2.7. Comparison of the $z = 0$ plane velocity distribution of wall-driven flow calculated using Equation (2.20) and Equation (2.26). The aspect ratio of the duct is (a) $w/h = 1$; (b) $w/h = 2$; (c) $w/h = 5$; (d) $w/h = 10$. 21

2.8. The normalized average velocity as a function of aspect ratio ($w/h$): (a) pressure-driven flows; (b) wall-driven flows. 22

3.1. (a) Primary surface design; (b) A perforated sheet with lubrication cavity. 25

3.2. Primary surface design with $A_{\text{cavity}}$ (surrounded by red dot-dashed line), $A_{\text{test}}$ (surrounded by black dashed line), $A_{\text{total}}$ (surrounded by blue solid line). 27

3.3. (a) Sketch of the laser micromachining system; (b) An example input file; (c) The visualization of the pattern described by (b). 28

3.4. Laser cut polyimide sheet bonded on the top of a cavity. The dimensions of the hole array and disassembled view have been presented in Figure 3.1. 30

3.5. Schematic of the model with general boundary conditions for (a) fully developed laminar flow in a duct; (b) fully developed laminar flow between two parallel plates. 31

3.6. Pressure loss estimation for no-slip and Navier-slip boundary. 32

3.7. Effective slip length computational procedure based on experimental measurements or theoretical results. 34

3.8. The flow cell design (top view and cross-section view). The dimensions of the polyimide sheet can be found in Figure 3.1. 36

3.9. Exploded view of the improved flow cell. The dimensions of the perforated sheet are given in Figure 3.1, and the flat sheets are the same size as the perforated one. All of the sheets are made of polyimide. 38

4.1. Laplace pressure $\Delta P_{\text{Laplace}}$ as a function of the protrusion angle $\theta$. 43
4.2. Gauge pressure as a function of time when the lubrication cavity was filled with water. \( P_A, P_B, P_C, \) and \( P_D \) represent the gauge pressure values that were measured by Sensor A, Sensor B, Sensor C, and Sensor D, respectively. 47

4.3. Gauge pressure as a function of time when the lubrication cavity was filled with air. \( P_A, P_B, P_C, \) and \( P_D \) represent the gauge pressure values that were measured by Sensor A, Sensor B, Sensor C, and Sensor D, respectively. 48

4.4. Schematic view of two parallel plates with different slip lengths \( (b_+, b_-) \) at both top and bottom walls, where \( H \) represents the distance between two parallel plates. 51

4.5. The drag reduction \( DR \) as a function of the mass flow rate \( \dot{m} \) for Case 1 \((d = 1.0 \text{ mm}, s = 1.2 \text{ mm})\) and Case 2 \((d = 1.0 \text{ mm}, s = 1.5 \text{ mm})\). 57

4.6. Schematic illustration of the air-water interface of entrapped air pockets. 58

4.7. The drag reduction \( DR \) as a function of the mass flow rate \( \dot{m} \) for Case 3 \((d = 1.0 \text{ mm}, s = 1.2 \text{ mm})\), Case 4 \((d = 0.8 \text{ mm}, s = 1.2 \text{ mm})\) and Case 5 \((d = 0.6 \text{ mm}, s = 1.5 \text{ mm})\). 62

4.8. The effective slip length \( b_{\text{eff}} \) as a function of the mass flow rate \( \dot{m} \) when the lubrication cavity is filled with air. 63

4.9. Schematic diagram of the full-scale model. 65

4.10. Schematic diagram of the unit cell model. 66

4.11. The pressure distribution along the flow direction. In this case, the cavity depth is 4 mm, and the uniform inlet velocity is 2.8842 m/s. 67

4.12. The pressure distribution along the flow direction at the different uniform inlet velocity. In this case, the cavity depth is 4 mm. 68

4.13. The pressure distribution along the flow direction computed with different cavity depth. In this case, the uniform inlet velocity is 2.8842 m/s. 69

4.14. The shape of the air-water interface in the unit cell simulation domain: (a) Convex; (b) Concave. The color bar refers to the velocity magnitude, which is given in meters per second. 70

4.15. The effective slip length \( b_{\text{eff}} \) as a function of protrusion angle \( \theta \). 71
4.16. Schematic diagram of the studied patterns, which are labeled from case 1 to case 12. In these cases, the no-slip area is marked in gray while the no-shear area is marked in white. Their geometry and layout are described by a unit cell (marked with the red dashed line) manner. (1) no-shear groove parallel to the flow direction; (2) no-shear groove transverse to the flow direction; (3) no-slip circle on a rectangular lattice; (4) no-shear circle on a rectangular lattice; (5) no-slip circle on a hexagonal lattice with vertical rows; (6) no-slip circle on a hexagonal lattice with horizontal rows; (7) no-slip hexagon on a hexagonal lattice with vertical rows; (8) no-slip hexagon on a hexagonal lattice with horizontal rows; (9) no-shear circle on a hexagonal lattice with vertical rows; (10) no-shear circle on a hexagonal lattice with horizontal rows; (11) no-shear hexagon on a hexagonal lattice with vertical rows; (12) no-shear hexagon on a hexagonal lattice with horizontal rows.

4.17. Slip length estimation for different patterns with a fixed porosity (75%) using the published analytical formula listed in Table 1 (red circle) and our CFD-based method with pressure-driven (blue filled circle) or wall-driven flow (black circle) assumption. The studied patterns are labeled from case 1 to case 12, and their geometry and layout have been illustrated and described in Figure 4.16.
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1. Relationship between $au + c\frac{\partial u}{\partial y} + d = 0$ Equation (2.1) and commonly used boundary conditions.</td>
<td>12</td>
</tr>
<tr>
<td>3.1. The parameters used for laser micromachining the polyimide sheets.</td>
<td>29</td>
</tr>
<tr>
<td>3.2. Relationship between $au + c\frac{\partial u}{\partial y} + d = 0$ (Equation (2.1)) and commonly used boundary conditions.</td>
<td>31</td>
</tr>
<tr>
<td>4.1. The performance of the syringe pump.</td>
<td>46</td>
</tr>
<tr>
<td>4.2. Summary of the cases experimentally investigated.</td>
<td>54</td>
</tr>
<tr>
<td>4.3. $Re$, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 1.</td>
<td>55</td>
</tr>
<tr>
<td>4.4. $Re$, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 2.</td>
<td>55</td>
</tr>
<tr>
<td>4.5. $Re$, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 3.</td>
<td>59</td>
</tr>
<tr>
<td>4.6. $Re$, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 4.</td>
<td>60</td>
</tr>
<tr>
<td>4.7. $Re$, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 5.</td>
<td>60</td>
</tr>
</tbody>
</table>
To my advisor, Professor Willis, who has taught me within and beyond the academia,
more than I can ever hope from a Ph.D. advisor,
and
to my parents, for their endless love, support and encouragement.
1.1 Motivation

Economic globalization has made today’s world more closely linked, where transportation (e.g. shipping, pipeline) has played an important role. Approximately over 60% (in air), 80% (underwater), and 100% (in pipeline) propulsive power is used to overcome the surface drag [1–3], so reducing surface drag will have a substantial impact economically and environmentally. As a consequence, there is always an interest in developing techniques or practices that can significantly reduce the surface drag, and numerous studies have been performed using either theoretical or experimental methods.

The motivation for this study is to develop a surface that has less surface drag and characterize its drag reduction performance, which then can be used to improve propulsion efficiency at intermediate Reynolds number (Re) flow. As emphasized by Karatay et al. [4], it is essential to manipulate the hydrodynamic boundary for drag reduction. The intuitive idea is to convert no-slip boundary conditions to slip or partial slip ones. To achieve that, the most promising approach is to replace the solid-liquid boundaries completely or partially with gas-liquid or liquid-liquid boundaries, because the shear stress is much smaller at the gas-liquid and liquid-liquid interfaces compared to that at solid-liquid interfaces. And the medium (gas/liquid) that reduces the shear on the boundaries is referred to as lubricant in this work.

In the current study, a surface that can effectively trap gas or liquid near the surface by its submillimeter textures is proposed, designed, fabricated, and characterized with a customized test configuration that includes a rectangular flow channel, lubricant (gas/liquid) cavity, a flow control pump, pressure transducers, and a computer used for test system control and data acquisition. An analytical model is developed to extract the representative quantities
that reflect the overall slip effects, which has been used to analyze the experimental data. In addition, numerical simulations (using COMSOL Multiphysics) were used to estimate the drag reduction performance of a given surface and understand the corresponding flow behaviors.

1.2 Research Background

1.2.1 Surfaces with drag reduction effect

For drag reduction, it is ideal to have a uniform gas layer between a working liquid flow and a solid surface. Nevertheless, this configuration is never practical, because of its thermodynamic instability. Alternatively, many attempts were made to produce gas-liquid interfaces at the boundaries utilizing superhydrophobic (SHPo) surfaces, which are commonly artificially created by introducing physical roughness (nano-scale or micro-scale structures) to chemically hydrophobic surfaces. In between the structures, pockets of gas could be formed, so that the drag experienced by the liquid flow over the SHPo surfaces is reduced [5–12].

However, gas pockets can be vulnerable, especially when there is a large pressure change across the gas-liquid interface [13,14]. Once the gas escapes from structures where it was trapped, a SHPo surface will lose all or part of its drag reduction effect [15]. Due to gas diffusion into the liquid, the lifetime of the gas pockets is limited. Emami et al. [16] proposed an analytical model to predict the longevity of the gas pockets. With help of Emami’s model, Xu et al. [17] successfully designed and fabricated a SHPo surface which was able to sustain gas pockets for a nearly infinite life time (> 50 days). This is a good example where an innovative theoretical work inspires an experimental breakthrough, which happens a lot in the frontier of science.

Another challenge is to restore the gas pockets after they collapse due to the liquid pressure. Lee and Kim [18,19] successfully demonstrated gas restoration on a specially designed SHPo surface with gas generation via electrolysis under water. As early as 2008, Carlborg et al. [20] coupled the gas pockets trapped in between the structures of a SHPo surface to a pneumatic pressure source, which allowed the researchers to actively control the gas pockets and investigate the drag with and without the presence of gas pockets. With
the same gas restoration principle, Vüllers et al. [21] created a nanostructured SHPo surface where the trapped gas pockets could be sustained for more than 4 hours at pressures up to 3.5 bar. Inspired by Carlborg’s and Vüllers’s works, Li et al. [15] fabricated a SHPo surface on a porous substrate, whose interconnected micropores connected the gas pockets trapped in between the roughness structures to an active air cavity. This design allowed air plastron restoration under water and showed the capability of drag reduction. Most of the time, the gas-liquid interface is not flat, because of the pressure difference across and the surface tension on it. The effect of gas-liquid interface’s shape on drag was reported by Steinberger [22], and then confirmed by Karatay et al. [4] in both experiments (where an actively controlled SHPo surface was used) and simulations (where a 2D model was solved with COMSOL).

Besides SHPo surfaces in which gases are trapped, lubrication-impregnated (LI) surfaces where lubricating liquids are impregnated, can also reduce the drag. For example, Solomon et al. [23] experimentally measured the drag reduction effect of their LI surface and compared it to a few previously published results obtained on LI surfaces.

1.2.2 Metric of surfaces with drag reduction effect

There are several metrics available to evaluate the drag reduction effects, such as effective slip length [4,23–30] and drag reduction (DR) [23,31–35]. The effective slip length (also known as the apparent slip length) is defined as [26]

\[ b_{\text{eff}} = \frac{(U_{\text{slip}})_{\text{wall}}}{\partial u / \partial y} \]  

(1.1)

where \((U_{\text{slip}})_{\text{wall}}\) is the average slip velocity at the wall, \(u\) is the streamwise velocity, and the \(y\)-axis is normal to the wall. As \(\frac{\partial u}{\partial y}\) \(\frac{\partial u}{\partial y}\) is related to the average shear stress \((\tau_{\text{wall}})\) via \(\tau_{\text{wall}} = \mu \frac{\partial u}{\partial y}\), the effective slip length \(b_{\text{eff}}\) can also be expressed in term of average shear stress at the wall. Therefore, if \((U_{\text{slip}})_{\text{wall}}\) and \(\tau_{\text{wall}}\) (or \(\frac{\partial u}{\partial y}\)) are given by experimental data or numerical model results, the effective slip length can be determined.
Drag reduction is defined in a relative manner i.e.

\[
\text{DR} = \frac{T_{\text{test}} - T_{\text{ref}}}{T_{\text{ref}}}
\]  

(1.2)

where \(T_{\text{test}}\) and \(T_{\text{ref}}\) could be torque, force, pressure loss, average shear stress, average velocity or flow rate, and the testing surface and reference surface are labeled by their subscripts. Thus, the definition of DR is closely related to the experiment or simulation configuration, so it is less general compared to effective slip length. However, DR is more intuitive to show the drag reduction effect, and hence widely used by researchers.

As summarized by Lee et al. [24], there are three experimental techniques to characterize slip effects. The first one is to measure the flow rate and pressure loss along the streamwise direction over the surface so that the effective slip length can be determined with the Navier model [25]. DR can also be calculated with Equation (1.2) from the measurements (flow rate or pressure loss) over the test surface (e.g. a surface with slip effect) and reference surface (e.g. a no-slip surface), which was used by Ou et al. [34] and Daniello et al. [35] to investigate the drag reduction effect in laminar and turbulent duct flow, respectively.
Figure 1.2: Experimental techniques used for the slip measurements and surface structures (both regular and random structures) probed by each [24].

The second technique requires measuring the velocity profile near the surface, and then the effective slip length can be evaluated according to its definition Equation (1.2). So, it is the only direct measurement technique among the three. The velocity profile is usually obtained by micron resolution particle image velocimetry (micro-PIV or µPIV), which can provide comprehensive information of the fluid velocity field near the surface. Therefore, this technique can be used to not only measure relatively weak slip effects [27–30], but also study the effects of the gas-liquid shape on the effective slip length [4].

The last technique refers to the ones measuring the force caused by the shear stress between liquid and adjacent solid surface. Only under a few specially designed experimental conditions, the relation between the force measurement and the effective slip length is available, such as Lee et al.’s configuration [36]. Consequently, this technique is more often used to measure DR, such as Hu et al.’s experiment [31,32]. In addition, the three techniques
described above can be used simultaneously. For instance, Hao et al. combined the first two techniques to find the effective slip length of their SHPo microchannel wall. Specifically, they measured the flow rate and pressure loss, and obtained the slip velocity at the SHPo wall using micro-PIV, from which the effect slip length can be computed with a modified series solution of a laminar duct flow [33].

1.2.3 Analytical model with Navier-slip conditions (Navier model)

The Navier-slip condition was first proposed by Navier in [37], which assumed that the tangential slip velocity at the wall is linearly proportional to the velocity gradient at the same place i.e.

\[ U_{\text{slip}} = b \frac{\partial u}{\partial y} \]  

(1.3)

where \( b \) is the linear coefficient, called Navier slip length. Solving models with Navier-slip conditions can reveal the relations among average velocity, pressure gradient, and Navier slip length, especially the analytical relation. Kashaninejad et al. [38] presented a solution for a fully developed laminar flow between two parallel Navier-slip plates, and the same author proposed a series solution for a fully developed laminar duct flow with Navier-slip boundaries [39]. In addition, Hao et al. [33] assumed a constant slip velocity on all duct walls, solved the fluid velocity field, and gave an expression for Navier slip length based on Equation 1.3. As proposed by Lauga and Stone [40], for a general flow with mixed slip/no-slip boundary conditions, the effective slip length is defined as the slip length of the flow that is driven by the same pressure gradient and has the same flow rate as the flow of the interest. Therefore, when the pressure gradient and flow rate are experimentally measured, the effective slip length can be solved from the analytical model with Navier-slip conditions.

Two-dimensional models [38] were widely used to determine the effective slip length via measuring flow rate and pressure drop [34]. In experiments, the flow rate and pressure drop are usually measured in a rectangular duct, which has two no-slip side walls, so that the no-slip side walls will contribute to the effective slip length estimation. In other words, the obtained effective slip length measures the combined effects of slip surfaces and no-slip side walls rather than the pure slip effect of the surfaces of interest. Therefore, Navier models for a laminar duct flow have received increasing attention recently. Two were reviewed in
the previous paragraph, but they all have their own shortcomings: the one proposed by Kashaninejad et al. [38] requires solving a transcendental equation for the coefficients used in their solution; the other one suggested by Hao et al. [33] requires all the duct walls to have the same slip effect, which is not always true. Thus, a more general 3D Navier model that can evaluate the effective slip length without interference from the no-slip sidewalls was developed in this work.

1.2.4 Analytical formulas for effective slip length

Predicting the effective slip length of a given surface structure is greatly desired, which is essential for surface design and its performance optimization without experimental measurements. However, analytical formulas are only available for several simple geometries summarized in Figure 1.3 and Equation (1.4). Thus, these geometries were widely used in SHPo and LI surface fabrication, while the application of more complex geometries is constrained by the absence of corresponding analytical formulas.

\[
b_{\text{eff}} = s \cdot \frac{\ln [\sec(\phi_g/2)]}{\pi} \tag{1.4a}
\]

\[
b_{\text{eff}} = s \cdot \frac{\ln [\sec(\phi_g/2)]}{2\pi} \tag{1.4b}
\]

\[
b_{\text{eff}} = s \cdot \left[ \frac{3}{16} \sqrt{\frac{\pi}{1 - \phi_g}} - \frac{3 \ln (1 + \sqrt{2})}{2\pi} \right] \text{ for } \phi_g > 0.30 \tag{1.4c}
\]

\[
b_{\text{eff}} = s \cdot [-A \ln (1 - \phi_g) + B] \text{ for } 0.25 < \phi_g < 0.78 \tag{1.4d}
\]

where \( s \) is the characteristic spacing (i.e. center-to-center distance), \( A = 0.134, B = -0.023 \), and \( \phi_g \) is void ratio, also known as the porosity of a surface.

For generic geometries, scaling laws and empirical formulas proposed by Ybert et al. [41] can give a rough estimation of the effective slip length, whereas numerical simulation should be used for a better prediction because more physics can be included in the simulation. Samaha et al. [26] assumed a flat no-shear gas-liquid interface and solved the fluid field with the finite-volume method. Karatay et al. [4] described a rigid circular no-shear gas-liquid
interface and used the finite-element method to solve the velocity field. Gao and Feng [42] introduced the diffuse-interface model to simulate the interface deformation due to the surface tension and liquid pressure, and the finite element method was used to compute the velocity field. In addition, Hyväluoma and Harting [43] studied the slip flow over the microbubble trapped in between the surface structures with the two-phase lattice Boltzmann method. Although numerical simulation yields better prediction and provides more information, the computational time usually makes it unsuitable for surface design and optimization.

Analytical formulas for effective slip length can not only reveal the dependence of effective slip length, but also reflect its variance due to different geometric configurations. For

Figure 1.3: The simple surface patterns and their corresponding formulas for theoretical predictions: (a) Grates parallel to the liquid flow, Equation (1.4a). (b) Grates transverse Equation (1.4b). (c) Posts, Equation (1.4c). (d) Holes, Equation (1.4d). [24]
instance, among the four simple patterns described in Figure 1.3, the effective slip length on surfaces with posts increases much more quickly compared to that on the surfaces with grates or holes, at high gas fraction [24]. Also, it is not yet known whether there exists a better design in addition to these four or not. Therefore, several numerical models were developed to systematically evaluate the effective slip length for different generic pattern, so that the optimal design will be distinguished.

1.3 Overview

In previous studies, the slip or partial slip conditions were created by SHPo or LI surfaces, which can introduce gas-liquid or liquid-liquid interfaces at boundaries. In engineering practice, actively controlled SHPo or LI surfaces is preferred, because of their robustness. However, turning a large surface (e.g. surface of the boat) into either SHPo or LI surfaces would be very challenging, because cleanroom techniques are required to fabricate micro-scale and nano-scale structures of SHPo or LI surfaces. To address these challenges, an actively controlled surface was developed that not only can provide slip or partial slip conditions, but also has a larger feature size so that conventional tools can be utilized for fabrication.

Chapter 2 introduces an analytical model for slip duct flow, which can be used to evaluate the effective slip length without interference from the no-slip sidewalls. Chapter 3 explains the design of the drag reduction surface and describes its fabrication procedures. The experimental test configuration is also described in this chapter. Chapter 4 is focused on discussing the results of experimental measurements and simulation results are used to explain the experimental findings. At last, Chapter 5 summarizes conclusions and discusses several interesting topics that could be immediate expansions of the current dissertation work.
2.1 Introduction

A variety of micro- and nano-manufacturing techniques have been developed that allow the fabrication of superhydrophobic (SHPo) surfaces [17,18,44,45]. Such engineered surfaces have allowed researchers to generate non-wetting surfaces for numerous applications such as self-cleaning, self-assembly, and bioengineering [24]. SHPo surfaces typically work by trapping gas in microscopic surface features, resulting in a composite surface, in which liquid contacts a solid surface only at certain locations, while the remainder of the surface is a liquid-gas interface. In the regions with liquid-gas contact, the shear stress is much lower than what would exist for a pure liquid-solid interface, resulting in slip flow behavior as described by the Navier slip condition [4,38–40]. The possibility of reduced shear stress has motivated a number of studies aimed at reducing drag on external surfaces [15] or pressure drop in internal flows [25,27,28,33,35]. A surface with different features (asperities, voids, etc.) could be represented by different effective boundary conditions (no-slip, partial slip, etc.), so quantifying the relationship between the general boundary conditions and flow behavior is important for engineering purposes.

Analytical models are essential for analyzing experimental results because analytical models are widely used to determine the characteristic quantities (e.g., the effective slip length) from measurements [4,25,33]. As proposed by Lauga and Stone [40], a composite surface adjacent to a flow can be represented by a surface with a slip length (i.e., Navier-slip length) that leads to the same flow rate under the same pressure gradient. Hence, this slip length is also known as the effective slip length.

If a model for such flows exists, the effective slip length could be expressed in terms of the pressure gradient and flow rate. Thus, when the pressure gradient and flow rate are
experimentally measured, the effective slip length can be determined. The desire for such models has driven multiple attempts at model formulation. Karatay et al. [4] applied the 2D Stokes model to a rectangular duct with Navier-slip conditions and deduced the expression for the effective slip length, in which the presence of sidewalls was completely ignored. Hao et al. [33] assumed a constant slip velocity on all duct walls, solved the fluid velocity field, and found an expression for slip length. In their assumptions, the no-slip sidewalls were replaced by the slip ones, so their prediction of the slip length is smaller than the measured amount. Kashaninejad et al. [38] presented a solution for a fully-developed laminar flow between two parallel Navier-slip plates, and the same author proposed a series solution for a fully-developed laminar duct flow with Navier-slip boundaries [39].

Laminar flow in ducts has been studied by many researchers, and the solutions for different shaped ducts, including the rectangular ones, were summarized in Shah and London’s book [46]. However, in the book, laminar duct flow with boundary conditions other than the no-slip ones remained undiscussed. In 2019, Kashaninejad [39] proposed a series solution for laminar duct flow with Navier-slip boundary conditions applied to all duct walls, the coefficients of which can only be approximately solved from the corresponding transcendental equations. Hence, Kashaninejad’s solution is not feasible for making measurements (e.g., effective slip length). In this work, an analytical model for fully-developed laminar flow in a rectangular duct is demonstrated. The duct has two opposing walls that experience no-slip conditions, while the other two walls have a general boundary condition that accounts for several possibilities: no-slip, no-shear, fixed shear, Navier-slip, and fixed velocity boundary conditions. The general boundary condition also accounts for the possibility of asymmetric boundaries, i.e., one slip and one no-slip. With this boundary condition configuration, the coefficients in this solution can be explicitly determined by the flow parameters, which make it feasible for making measurements.

2.2 Methods

2.2.1 General boundary condition

The boundary condition of a fluid flow problem usually can be expressed in terms of velocity, \( u \), or velocity gradient, \( \partial u / \partial y \), such as no-slip, \( u = 0 \), free shear, \( \partial u / \partial y = 0 \), and
Navier slip, \( u = b(\partial u/\partial y) \), where \( b \) is the slip length. Thus, we propose a general expression for boundary conditions

\[
a u + c \frac{\partial u}{\partial y} + d = 0
\]  

(2.1)

where \( a, c, d \) are parameters that allow Equation (2.1) to represent all of the commonly used boundary conditions, which have been summarized in Table 2.1. For the Navier-slip boundary condition, \( c \) is related to slip length \( b \) following the sign convention proposed by Kashaninejad [38] i.e. \( c = b \) on the top wall and \( c = -b \) on the bottom wall.

Table 2.1: Relationship between \( au + c\frac{\partial u}{\partial y} + d = 0 \) Equation (2.1) and commonly used boundary conditions.

| \( a = 1, c = 0, d = 0 \) | No-slip |
| \( a = 1, c = 0, d \neq 0 \) | Fixed velocity |
| \( a = 0, c = 1, d \in \mathbb{R} \) | No-shear or fixed shear |
| \( a = 1, c \neq 0, d = 0 \) | Navier slip |
| \( a \neq 0, c \neq 0, d = 0 \) | Mixed boundary condition |

2.2.2 Fully-developed laminar flow in a rectangular duct

In this section, a fully developed, incompressible, parallel, steady, laminar duct flow is investigated with constant physical properties and boundary conditions as illustrated schematically in Figure 2.1. Based on these assumptions, the governing equation can be obtained by simplifying the Navier-Stokes (N-S) equations as

\[
\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} = -\frac{\Delta p}{\mu L}
\]

(2.2)

where \( u \) denotes the \( x \)-component of the flow velocity, \( \mu \) is the dynamic viscosity, and \(-\Delta p/L\) represents the pressure gradient along the \( x \)-direction. The no-slip boundary condition is applied to the left and right walls \((z = \pm w)\) of the duct, while the general boundary condition
proposed in Equation (2.1) is imposed on the top and bottom walls. In order to allow for asymmetric boundary conditions, Equation (2.1) is re-written as

\[ a_\pm u + c_\pm \frac{\partial u}{\partial y} + d_\pm = 0 \]  

(2.3)

where the subscript ‘+’ and ‘-’ are used to denote the top and bottom wall, respectively.

Figure 2.1: Schematic of the model: A rectangular duct with a height of \(2h\) and a width of \(2w\), with fully-developed, incompressible, parallel, steady, laminar flow along the \(x\)-direction. Equation (2.1) is applied to the surfaces at \(y = \pm h\).

Depending on the assumed pressure gradient and the boundary conditions imposed on top and bottom walls, the fluid can be driven by a pressure gradient, wall movement, or a combination of the two mechanisms. Therefore, the analytical solution governed by Equation (2.2) and solved with boundary conditions described by Equation (2.3), should be able to describe any behavior of fully developed laminar flow in a duct with at least two no-slip walls.

To satisfy the no-slip conditions prescribed on the side-walls \((z = \pm w)\), the solution of Equation (2.2) can take the following form

\[ u(y, z) = \sum_{n=0}^{\infty} A_n(y) \cos(\alpha_n z) \]  

(2.4)
where $\alpha_n = (2n+1)\pi/(2w)$, and $A_n(y)$ is the coefficient of $\cos(\alpha_n z)$ and a function of $y$ only. Then, $-\Delta p/\mu L$ can be expressed in terms of $\cos(\alpha_n z)$ as

$$-\frac{\Delta p}{\mu L} = \sum_{n=0}^{\infty} B_1 n(y) \cos(\alpha_n z)$$  \hspace{1cm} (2.5)$$

where $B_1 = \frac{4\Delta p}{\mu L (2n+1)\pi}$. Substituting Equation (2.4) and Equation (2.5) into Equation (2.2), yields

$$\sum_{n=0}^{\infty} \left[ A''_n(y) - \alpha_n^2 A_n(y) \right] \cos(\alpha_n z) = -\frac{4\Delta p}{\mu L} \sum_{n=0}^{\infty} \frac{(-1)^n}{(2n+1)^n \pi} \cos(\alpha_n z)$$  \hspace{1cm} (2.6)$$

For each value of $n$, the coefficient of $\cos(\alpha_n z)$ on each side of the equation is identical and yields an ordinary differential equation (ODE)

$$A''_n(y) - \alpha_n^2 A_n(y) = -\frac{4\Delta p}{\mu L} \frac{(-1)^n}{(2n+1)^n \pi}$$  \hspace{1cm} (2.7)$$

which has a solution given by

$$A_n(y) = C_n \cosh(\alpha_n y) + D_n \sinh(\alpha_n y) + \beta_n$$  \hspace{1cm} (2.8)$$

where $\beta_n = \frac{4\Delta p}{\mu L (2n+1)\pi} / \alpha_n^2$ is the inhomogeneous solution of Equation (2.8). Hence, $\partial A_n(y)/\partial y$ can be expressed by

$$\frac{\partial A_n(y)}{\partial y} = C_n \alpha_n \sinh(\alpha_n y) + D_n \alpha_n \cosh(\alpha_n y)$$  \hspace{1cm} (2.9)$$

Note that the coefficient $C_n$ and $D_n$ used in Equation (2.8) and Equation (2.9) can be determined with the boundary conditions. Specifically, the general boundary conditions prescribed on top ($y = h$) and bottom ($y = -h$) walls should be used. Hence, to accommodate Equation (2.6), the form of boundary conditions needs to be revisited.
First, \( d_+ \), and \( d_- \) are expressed in terms of \( \cos(\alpha_n z) \)

\[
d_+ = \sum_{n=0}^{\infty} B_{2n} \cos(\alpha_n z) \tag{2.10}
\]

\[
d_- = \sum_{n=0}^{\infty} B_{3n} \cos(\alpha_n z) \tag{2.11}
\]

where \( B_{2n} = d_+ \delta_n \), \( B_{3n} = d_- \delta_n \), and

\[
\delta_n = \frac{4}{(2n + 1)\pi} (-1)^n \tag{2.12}
\]

Substituting Equation (2.4), Equation (2.10), Equation (2.11) and \( y = \pm h \) into Equation (2.3), and collecting the coefficients in front of \( \cos(\alpha_n z) \) for each value of \( n \), yields

\[
a_+ [A_n(y)] + \left[ \frac{\partial A_n(y)}{\partial y} \right] + d_+ \delta_n = 0 \tag{2.13}
\]

\[
a_- [A_n(y)] + \left[ \frac{\partial A_n(y)}{\partial y} \right] + d_- \delta_n = 0 \tag{2.14}
\]

Then, substituting Equation (2.8) and Equation (2.9) into Equation (2.13) and Equation (2.14), respectively, and simplifying yields two equations for coefficients \( C_n \) and \( D_n \)

\[
\phi_+ C_n + \psi_+ D_n + \chi_+ = 0 \tag{2.15}
\]

\[
\phi_- C_n + \psi_- D_n + \chi_- = 0 \tag{2.16}
\]

where

\[
\phi_+ = a_+ \cosh(\alpha_n h) + c_+ \alpha_n \sinh(\alpha_n h)
\]

\[
\phi_- = a_- \cosh(\alpha_n h) - c_- \alpha_n \sinh(\alpha_n h)
\]

\[
\psi_+ = a_+ \sinh(\alpha_n h) + c_+ \alpha_n \cosh(\alpha_n h)
\]

\[
\psi_- = -a_- \sinh(\alpha_n h) + c_- \alpha_n \cosh(\alpha_n h)
\]

\[
\chi_+ = a_+ \beta_n + d_+ \delta_n
\]

\[
\chi_- = a_- \beta_n + d_- \delta_n
\]

Therefore, the coefficient \( C_n \) and \( D_n \) can be found by solving Equation (2.15) and Equa-
tion (2.16) numerically or analytically. To give more mathematical or physical insight, the analytical expression of \( C_n \) and \( D_n \) are explicitly given by

\[
C_n = \frac{[2a_+a_- \sinh (\alpha_n h) + (a_-c_- - a_+c_+) \alpha_n \cosh (\alpha_n h)] \beta_n}{\sinh (2\alpha_n h) (c_+c_-\alpha_n - a_+a_-) + \cosh (2\alpha_n h) \alpha_n (a_+c_- - a_-c_+)} + \frac{(a_+d_- + a_-d_+) \delta_n \sinh (\alpha_n h) + (c_+d_- - c_-d_+) \delta_n \alpha_n \cosh (\alpha_n h)}{\sinh (2\alpha_n h) (c_+c_-\alpha_n - a_+a_-) + \cosh (2\alpha_n h) \alpha_n (a_+c_- - a_-c_+)}
\]  

(2.18)

\[
D_n = -\frac{[(a_-c_+ + a_+c_-) \alpha_n \sin (\alpha_n h)] \beta_n}{\sinh (2\alpha_n h) (c_+c_-\alpha_n - a_+a_-) + \cosh (2\alpha_n h) \alpha_n (a_+c_- - a_-c_+)} + \frac{(c_+d_- + c_-d_+) \delta_n \alpha_n \sinh (\alpha_n h)}{\sinh (2\alpha_n h) (c_+c_-\alpha_n - a_+a_-) + \cosh (2\alpha_n h) \alpha_n (a_+c_- - a_-c_+)}
\]  

(2.19)

So far, all the coefficients needed to evaluate Equation (2.4) have been obtained. Therefore, the expressions for velocity profile, velocity gradient in \( y \) and \( z \), volume flow rate, and average velocity can be determined as

\[
u(y, z) &= \sum_{n=0}^\infty A_n(y) \cos (\alpha_n z) = \sum_{n=0}^\infty [C_n \cosh (\alpha_n y) + D_n \sinh (\alpha_n y) + \beta_n] \cos (\alpha_n z)  
\]

(2.20)

\[
\frac{\partial u(y, z)}{\partial y} &= \sum_{n=0}^\infty \frac{\partial A_n(y)}{\partial y} \cos (\alpha_n z) = \sum_{n=0}^\infty [C_n \sinh (\alpha_n y) + D_n \cosh (\alpha_n y)] \alpha_n \cos (\alpha_n z)  
\]

(2.21)

\[
\frac{\partial u(y, z)}{\partial z} &= \sum_{n=0}^\infty A_n(y) \frac{\partial \cos (\alpha_n z)}{\partial z} = \sum_{n=0}^\infty [C_n \cosh (\alpha_n y) + D_n \sinh (\alpha_n y) + \beta_n] (-\alpha_n) \sin (\alpha_n z)  
\]

(2.22)

\[
Q = \int_{-w-h}^\infty \int_{-h}^n u(y, z) dy dz = \sum_{n=0}^\infty \left\{ C_n \frac{1}{\alpha_n} [2 \sinh (\alpha_n h)] + 0 + 2h\beta_n \right\} \frac{1}{\alpha_n} [2(-1)^n]  
\]

(2.23)
\[ \bar{u} = \frac{Q}{(2w)(2h)} = \frac{1}{4wh} \sum_{n=0}^{\infty} \left\{ C_n \frac{1}{\alpha_n} \left[ 2 \sinh (\alpha_n h) \right] + 0 + 2h \beta_n \right\} \frac{1}{\alpha_n} [2(-1)^n] \quad (2.24) \]

Since the series terms in Equation (2.20) - Equation (2.24) decrease as \( n \to \infty \), good accuracy can be obtained even if the summations are truncated at some finite number (\( N \)). The selection of \( N \) depends on both the geometry of the duct and the pressure gradient, so we need to plot the series term as a function of its index, and then determine how many terms are needed. In general, a larger pressure gradient and higher aspect ratio (\( w/h \)) require more terms to be included.

2.2.3 Fully-developed laminar flow between two parallel plates

In this section, a fully developed incompressible laminar flow between two parallel plates \((z \to \pm \infty)\) is solved with the general boundary condition shown in Figure 2.2. Assuming steady flow and constant physical properties, the governing equation can be obtained as

\[ \frac{\partial^2 u}{\partial y^2} = -\frac{\Delta p}{\mu L} \quad (2.25) \]

where \( u \) denotes the \( x \)-component of the flow velocity, \( \mu \) is the dynamic viscosity, and \(-\Delta p/L\) represents the pressure gradient along \( x \)-direction.

\[
\begin{align*}
& a_+ u + c_+ \frac{\partial u}{\partial y} + d_+ = 0 \\
& a_- u + c_- \frac{\partial u}{\partial y} + d_- = 0
\end{align*}
\]

Figure 2.2: Schematic of the model: Two parallel plates with a spacing of \( 2h \), with fully-developed, incompressible, steady, laminar flow along the \( x \)-direction. Equation (2.1) is applied to the walls at \( y = \pm h \).
Integrating Equation (2.25) twice yields the expression for \( u \)

\[
u(y) = \frac{1}{2} \left( -\frac{\Delta p}{\mu L} \right) y^2 + \xi y + \zeta \tag{2.26}
\]

where \( \xi \) and \( \zeta \) are constants of integration, which can be determined by the boundary conditions imposed on the top and bottom walls. Based on Equation (2.26), the velocity gradient in \( y \) and the average velocity can be determined as

\[
\frac{\partial u(y)}{\partial y} = \left( -\frac{\Delta p}{\mu L} \right) y + \xi \tag{2.27}
\]

\[
\bar{u} = \frac{1}{2h} \int_{-h}^{h} u(y) \, dy = \frac{1}{2h} \left[ \frac{1}{3} \left( -\frac{\Delta p}{\mu L} \right) h^3 + 2\zeta h \right] \tag{2.28}
\]

where \( \xi \) and \( \zeta \) can be determined from the boundary conditions i.e.

\[
\zeta = \frac{a_+ \left\{ \left( -\frac{\Delta p}{\mu L} \right) \left[ a_- \left( \frac{1}{2} \right) h^2 - c_- h \right] + d_- \right\} - a_- \left\{ \left( -\frac{\Delta p}{\mu L} \right) \left[ a_+ \left( \frac{1}{2} \right) h^2 + c_+ h \right] + d_+ \right\}}{(a_+ h + c_+) a_- - (-a_- h + c_-) a_+} \tag{2.29}
\]

\[
\zeta = -\frac{(a_+ h + c_+) \left\{ \left( -\frac{\Delta p}{\mu L} \right) \left[ a_- \left( \frac{1}{2} \right) h^2 - c_- h \right] + d_- \right\} - (-a_- h + c_-) \left\{ \left( -\frac{\Delta p}{\mu L} \right) \left[ a_+ \left( \frac{1}{2} \right) h^2 + c_+ h \right] + d_+ \right\}}{(a_+ h + c_+) a_- - (-a_- h + c_-) a_+} \tag{2.30}
\]

\section*{2.3 Results}

The derived analytical solutions can describe the behaviors of the flows of interest by providing the normalized velocity distribution. In this section, the theoretical models were used to analyze fully-developed laminar slip flow in a rectangular duct. For simplicity but without loss of generality, the duct has two no-slip side walls and a top wall that experiences no-slip conditions, while the bottom wall has several possibilities: no-slip, no-shear, Navier-slip \((b = 50 \, \mu m)\), and fixed velocity \((u = 1.0 \, m/s)\) boundary conditions.

We defined the normalized velocity as \( u/u_{2D, \text{no-slip, max}} \) for pressure-driven flow, where \( u_{2D, \text{no-slip, max}} \) is the maximum velocity of the pressure-driven flow between two no-slip parallel
plates, and it is computed by

\[ u_{2D,\text{no-slip, max}} = -\left( -\frac{\Delta p}{L} \right) \frac{h^2}{2\mu} \]  \hspace{1cm} (2.31)

where \( \mu \) is the dynamic viscosity, and \(-\Delta p/L\) represents the pressure gradient along the \( x \)-direction. For wall-driven flows the velocity is normalized as \( u/u_{\text{wall}} \), where \( u_{\text{wall}} \) is the speed of the moving wall. Note that \( u_{\text{wall}} \) is also the maximum velocity of wall-driven flow. The coordinates in the \( z \)-direction and the \( y \)-direction are normalized using half of the channel width \( w \) and the half of the channel height \( h \) (i.e., the half of the spacing between two parallel plates in 2D cases). Moreover, the pressure gradient is \( 1 \times 10^4 \) Pa/m for all pressure-driven flows; the moving wall speed is 1.0 m/s for all wall-driven flows.

Figure 2.3: Comparison of the \( z = 0 \) plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is \( w/h = 1 \). (a) no-slip, Navier-slip (\( b = 50 \) \( \mu \)m), and fixed velocity (\( u = 1.0 \) m/s); (b) no-shear.
Figure 2.4: Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 2$. (a) no-slip, Navier-slip ($b = 50 \, \mu m$), and fixed velocity ($u = 1.0 \, m/s$); (b) no-shear.

Figure 2.5: Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 5$. (a) no-slip, Navier-slip ($b = 50 \, \mu m$), and fixed velocity ($u = 1.0 \, m/s$); (b) no-shear.
Figure 2.6: Comparison of the $z = 0$ plane velocity distribution of pressure-driven flow calculated using Equation (2.20) and Equation (2.26). For the duct flow, the aspect ratio is $w/h = 10$. (a) no-slip, Navier-slip ($b = 50 \, \mu m$), and fixed velocity ($u = 1.0 \, \text{m/s}$); (b) no-shear.

Figure 2.7: Comparison of the $z = 0$ plane velocity distribution of wall-driven flow calculated using Equation (2.20) and Equation (2.26). The aspect ratio of the duct is (a) $w/h = 1$; (b) $w/h = 2$; (c) $w/h = 5$; (d) $w/h = 10$. 
Figures 2.3-2.7 illustrate the variations of the normalized velocity distribution at \( z = 0 \) with respect to the aspect ratio, i.e., Equation (2.20) for duct flow. In all cases, the summation in the series solution is truncated at \( n = 210 \), which is needed by the case with the highest aspect ratio (\( \sim 100 \)) to provide a good accuracy. In the same figures, the velocity profiles predicted by the 2D analytical formula, i.e., Equation (2.26) for parallel plates are also presented as references. As the aspect ratio increases, the velocity distribution at \( z = 0 \) becomes more similar to the velocity profile given by the 2D analytical formula, which indicates the side-wall effects become negligible. Note that the 2D analytical formula corresponds to the limiting case of the duct flow when the aspect ratio approaches infinity.

The no-slip side-wall effects depend not only on the aspect ratio but also on boundary conditions imposed on the top and bottom wall. With the no-shear boundary condition at the bottom wall, the differences between the 2D velocity profile and the velocity distribution of the duct flow at \( z = 0 \) are still noticeable when \( w/h = 5.0 \), but the differences are negligible in the other cases.

The no-slip side-wall effects can also be reflected by the normalized average velocity, which is defined by \( \bar{u}/u_{2D,\text{no-slip},\text{max}} \) for pressure-driven flow, and \( \bar{u}/u_{\text{wall}} \) for wall-driven flow. The variance of the normalized average velocity as a function of the aspect ratio for several boundary conditions is illustrated in Figure 2.8. As the aspect ratio increases, the normalized
average velocity predicted for the duct flow model approaches the normalized average velocity given by the 2D analytical formula. When the aspect ratio is less than 10, the 2D model will predict a much larger normalized average velocity than that predicted by the duct flow model. So it will cause a significant deviation if the 2D model is used to analyze a rectangular channel with an aspect ratio of less than 10. When the aspect ratio increases, the differences between the normalized average velocity predicted by the 2D models and the duct flow model become smaller and negligible. In this case, the 2D model can be used for simplicity with acceptable error.

2.4 Conclusion

Rectangular channels are widely used in lab-on-a-chip and microfluidic devices. Due to the lack of an analytical model for duct slip flow, the no-slip side walls’ effects are usually neglected [4]. In this work, an analytical solution for duct flow with general boundary conditions was derived, allowing the side-wall effects to be included in duct flow analysis. The flow between two parallel plates was revisited with general boundary conditions, and a comprehensive solution was obtained that covers all the possibilities. For both models, the velocity field, the velocity gradient, the volume flow rate, and the average velocity were deduced.

Comparisons in normalized velocity profile and normalized average velocity were presented. When the aspect ratio is less than 10, the normalized velocity profile and normalized average velocity predicted by the duct model will significantly differ from that predicted by the 2D model due to the side-wall effects. However, the excellent agreement between the duct model analysis and the 2D model analysis for a very large aspect ratio ($w/h \approx 100$) indicates that the 2D model is accurate in this regime.
3.1 Surface design and fabrication

3.1.1 The design of a surface that can entrap gas or liquid

Gas or liquid can be entrapped within holes or in between posts, which distinguish two types of surfaces i.e. hole-type and post-type. Both hole-type design and post-type design are widely used to achieve gas/liquid entrapment on surfaces, e.g. superhydrophobic surfaces and lubrication surfaces. The parameters that are commonly used to describe such surfaces are feature size, spacing between features, and the void ratio (or gas/liquid fraction). The void ratio and gas fraction both mathematically equal surface porosity in this scenario. However, the term "void ratio" only relates to the geometry dimensions of the surface, and the "gas/liquid fraction" reflects the presence of the gas/liquid-liquid interface.

Compared with the hole-type surfaces, the entrapped gas/liquid are interconnected and less robust. As a consequence, the hole-type design was selected to develop the primary design of the surface that can entrap gas or liquid. For the hole-type surfaces, also known as the perforated surfaces, the porosity is more commonly used instead of gas/liquid fraction. Different expressions for porosity can be used under different scenarios. To avoid ambiguity and assist analysis and discussion, the concept of the porosity is subdivided into three categories, namely local porosity, $\phi_{\text{local}}$, cavity-area porosity, $\phi_{\text{cavity}}$, test-area porosity, $\phi_{\text{test}}$, and total porosity, $\phi_{\text{total}}$, which will be discussed later in this subsection.

As revealed by Ybert et al. [41], a higher gas fraction will lead to a larger effective slip length, i.e. less surface friction, because more of the surface area is replaced by the gas-liquid or liquid-liquid interfaces. Therefore, one of the key points of the primary design is to have a larger porosity so that more gas can be trapped.

From theoretical work done by Emami et al. [47,48], the shape of holes does have an im-
impact on the threshold liquid pressure that the gas-liquid or liquid-liquid interfaces entrapped in the holes can hold, and surfaces with circular holes can entrap gas pockets stably under high liquid pressure. So, a surface with dense-packed (i.e. hexagonal lattice) circular holes is selected as the primary design shown in Figure 3.1 (a), which can not only provide high gas fraction but also resist high liquid pressure. The dimensions and arrangement of one of the surface designs are also labeled in Figure 3.1 (b).

As mentioned earlier, porosity is one of the key parameters that describe hole-type surfaces and is subdivided into three categories: local porosity, cavity-area porosity, test-area porosity, and total porosity, given by

$$\phi_{\text{local}} = \frac{\sqrt{3}\pi}{6} \cdot \frac{d^2}{s^2}$$  \hspace{1cm} (3.1a)$$

$$\phi_{\text{cavity}} = \frac{N \cdot \pi \cdot \frac{d^2}{4}}{A_{\text{cavity}}}$$  \hspace{1cm} (3.1b)$$

$$\phi_{\text{test}} = \frac{N \cdot \pi \cdot \frac{d^2}{4}}{A_{\text{test}}}$$  \hspace{1cm} (3.1c)$$

$$\phi_{\text{total}} = \frac{N \cdot \pi \cdot \frac{d^2}{4}}{A_{\text{total}}}$$  \hspace{1cm} (3.1d)$$
\[ \phi_{\text{cavity}} A_{\text{cavity}} = \phi_{\text{test}} A_{\text{test}} = \phi_{\text{total}} A_{\text{total}} \quad (3.1e) \]

where \( d \) is the diameter of the holes, \( s \) is the center-to-center spacing between holes, and \( N \) is the number of holes. \( A_{\text{cavity}} \) (surrounded by red dot-dashed line), \( A_{\text{test}} \) (surrounded by black dashed line), \( A_{\text{total}} \) (surrounded by blue solid line) are labeled correspondingly in Figure 3.2. According to Equation (3.1a), local porosity is determined by the pattern itself and has nothing to do with the size of the area that the pattern covers. Thus, the local porosity is commonly used to discuss the pattern-dependent effects. As for the other three porosities, they are proposed to incorporate the real-world configuration. For example, two perforated sheets of the same size have the same hole diameter and spacing, i.e. the same local porosity, but they can have a different number of holes, i.e. total porosity. The perforated sheet with more holes is expected to experience less drag because it has less no-slip area. The holes on the perforated sheets are designed to trap air bubbles, so they must be located within the area that the cavity can cover (surrounded by the red dot-dashed rectangle in Figure 3.2) to be connected to the cavity. The cavity porosity is the highest porosity that can be achieved for a given pattern in a real application. However, some extra white space outside of the perforated region (within the solid blue rectangle but outside the red dot-dashed rectangle in Figure 3.2) is needed for the installation or bonding process. In experiments, the pressure loss is measured over the test region surrounded by the black dashed rectangle in Figure 3.2, so the test porosity is introduced to include the effect of the no-slip area (within the black dashed rectangle but outside the red dot-dashed rectangle). In summary, local porosity and test-area porosity are used more often, because the former represents the characteristics of the pattern itself, and the latter reflects the experimental conditions.

For the pattern illustrated in Figure 3.2, \( N = 200 \), \( d = 1.0 \) mm, \( s = 1.2 \) mm, the width and the length of \( A_{\text{cavity}} \) is 8.0 mm and 35.0 mm, the width and the length of \( A_{\text{test}} \) is 11.9 mm and 35.0 mm, and the width and the length of \( A_{\text{total}} \) is 11.9 mm and 40.0 mm respectively. Note that the width and length of \( A_{\text{cavity}}, A_{\text{test}}, \) and \( A_{\text{total}} \) are fixed in all surface designs.

According to the information given, the local porosity (i.e. porosity within the hexagonal lattice), cavity-area porosity, test-area porosity, and total porosity of the pattern shown in Figure 3.2 are 62.98\%, 56.10\%, 37.40\%, and 33.00\%, respectively.
Figure 3.2: Primary surface design with $A_{\text{cavity}}$ (surrounded by red dot-dashed line), $A_{\text{test}}$ (surrounded by black dashed line), $A_{\text{total}}$ (surrounded by blue solid line).

Since there is no analytical formula available for the surface with holes on a hexagonal lattice, Equation (1.4d) (holes on a rectangular lattice) is used to roughly estimate the effective slip length. With the local porosity of 62.98%, Equation (1.4d) yields an effective slip length of 132.2 $\mu$m, which is called the estimated effective slip length of the primary design.

To supply gas/liquid to structures (i.e. into holes or in between the posts) and control the entrapped gas/liquid, a reservoir that contains gas/liquid needs to be connected to the features, the implementation of which is shown in Figure 3.1 (b). The perforated sheet can be bonded on the top of a cavity with a layer of laser-cut double-sided tape. Hence, the gas/liquid pockets entrapped in the hole array can be actively controlled via alternating the pressure in the cavity.

3.1.2 Generic pattern fabrication via laser micromachining system

The perforated sheet and the double-sided tape are processed with a laser micromachining system. The laser micromachining system consists of a laser, a 3-axis motion stage controlled by a computer, and necessary optical components as shown in Figure 3.1 (a). The laser is a diode-pumped Q-switched Nd:YLF laser with an output wavelength of $\lambda = 349$ nm, and pulse width (FWHM) $\tau < 5$ ns. The 3-axis motion stage is actuated by two LTA-HS
motorized actuators (x-axis and y-axis) and one LTA-HS motorized actuator (z-axis). These three motorized actuators are controlled by a Newport ESP-300 motion controller. As shown in Figure 3.1 (a), the laser is focused at a fixed location and the sample that needs to be processed is mounted on and moved with the 3-axis stage. Then, a clean laser cut is left on the sample.

Generic pattern fabrication requires sophisticated control of the laser and the motion stage, so a control software was developed. The software controlled the laser and stage according to the input file where the stage motion path and laser operation parameters are described. In addition, a MATLAB script was created to visualize the stage motion (in the x-y plane) described in the file. Figure 3.3 (b) shows a typical input file, whose visualization is demonstrated in Figure 3.3 (c). The detailed protocol for creating the fabrication input files is documented in Appendix A.

Figure 3.3: (a) Sketch of the laser micromachining system; (b) An example input file; (c) The visualization of the pattern described by (b).
Kapton HN Polyimide (PI) sheets (125 µm thick) were used for surface fabrication because they can be easily laser-ablated due to their high absorption nature to ultraviolet light and decomposition into gaseous products. With the described laser micromachining system, various patterns can be cut on the polyimide sheets, as long as the pattern has been properly described in the fabrication input file according to the protocol presented in Appendix A. Similarly, the double-sided tape can also be cut, but with a different setting. The key parameters for laser-cutting polyimide sheets and double-sided tape have been summarized in Table 3.1.

<table>
<thead>
<tr>
<th>Materials</th>
<th>Pulse energy</th>
<th>Frequency</th>
<th>Speed</th>
<th>Acceleration</th>
<th>Pass</th>
</tr>
</thead>
<tbody>
<tr>
<td>Polyimide</td>
<td>108 µJ</td>
<td>1500 Hz</td>
<td>0.3 mm/s</td>
<td>10.0 mm/s²</td>
<td>1</td>
</tr>
<tr>
<td>Double-sided tape</td>
<td>118 µJ</td>
<td>1300 Hz</td>
<td>0.3 mm/s</td>
<td>20.0 mm/s²</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 3.4 illustrates the primary design that has been successfully implemented, where the dense-packed (i.e. hexagonal lattice) circular holes are fabricated on a polyimide sheet, and installed on the top of the cavity with a layer of the double-sided tape cut into the needed shape. The regions that were covered by the side of the lid of the flow cell are marked as shade. The direction of the water flow and gas/liquid supplying path is labeled with blue arrows and a green arrow, respectively. The dimensions and layout of the hole array and disassembled view have been presented in Figure 3.1.
3.2 Effective slip length estimation and measurement

3.2.1 Analytical models for a fully developed laminar flow

The fully developed laminar flow between two parallel plates and in a duct was investigated with the configuration illustrated in Figure 3.5, where a general boundary condition is applied at the top and bottom walls, and flow is in the $x$-direction. As summarized in Table 3.2, the general boundary condition can represent several possibilities: no-slip, Navier slip, no-shear, fixed shear, fixed velocity, and mixed boundary condition.

With the 2D model described in Figure 3.5 (a), the expression of the average velocity has been given in Equation (2.28). Similarly, with the 3D configuration illustrated in Figure 3.5 (b), the average velocity of the duct flow has been deduced in Equation (2.24). Using Equation (2.28) and Equation (2.24), the average velocity can be calculated with the dimensions of the configuration, pressure gradient, and boundary conditions. Conversely, when the pressure gradient is unknown but the average velocity is given, the pressure gradient can be determined. For the 2D configuration, the pressure gradient can be found either analyt-
Figure 3.5: Schematic of the model with general boundary conditions for (a) fully developed laminar flow in a duct; (b) fully developed laminar flow between two parallel plates.

Table 3.2: Relationship between $a u + c \frac{\partial u}{\partial y} + d = 0$ (Equation (2.1)) and commonly used boundary conditions.

<table>
<thead>
<tr>
<th>Condition</th>
<th>Equation</th>
</tr>
</thead>
<tbody>
<tr>
<td>No-slip</td>
<td>$a = 1, c = 0, d = 0$</td>
</tr>
<tr>
<td>Fixed velocity</td>
<td>$a = 1, c = 0, d \neq 0$</td>
</tr>
<tr>
<td>No-shear or fixed shear</td>
<td>$a = 0, c = 1, d \in \mathbb{R}$</td>
</tr>
<tr>
<td>Navier slip</td>
<td>$a = 1, c \neq 0, d = 0$</td>
</tr>
<tr>
<td>Mixed boundary condition</td>
<td>$a \neq 0, c \neq 0, d = 0$</td>
</tr>
</tbody>
</table>

In contrast, we cannot explicitly or iteratively from Equation (2.28), whereas, for the 3D configuration, it can only be solved from Equation (2.24) with the iterative method. Due to the simplicity of the 2D solution, it was widely used to analyze the flow in a high aspect ratio (>100) rectangular duct, where the side wall effects can be ignored. To take the side wall effects into account, the 3D solution should be used in duct flow analysis. With the 2D and 3D solutions, the pressure gradient can be solved at different average velocity $\bar{u}$ so that the corresponding pressure loss $\Delta p$ over a streamwise distance $L$ at a different mass flow rate $\dot{m}$ can be obtained. Specifically, the pressure loss $\Delta p$ over a streamwise distance $L$ is calculated by multiplying the pressure gradient with the distance $L$, and the mass flow rate is computed.
with $\dot{m} = \rho \bar{u} A$, where $\rho$ is the density of the fluid, $A$ is the cross-section area of the duct.

Figure 3.6: Pressure loss estimation for no-slip and Navier-slip boundary.

In Figure 3.6, both 2D and 3D solutions are used to calculate the pressure loss as a function of mass flow rate for a duct flow, where a rectangular duct with height $2h = 0.8$ mm and width $2w = 12.0$ mm was considered, and $L = 35$ mm was assumed. The duct can either have four no-slip walls (solid lines in Figure 3.6) or have a Navier-slip bottom wall ($b = 132.2$ µm) while the other three were no-slip (dashed lines in Figure 3.6). To illustrate the slip effects on the pressure loss, the choice of Navier slip length $b$ can be arbitrary. Here,
$b = 132.2 \, \mu m$ was chosen, which is at the same order of magnitude as the estimated effective slip length of the primary design. Note that $L = 35 \, \text{mm}$ is the distance between two pressure probe ports that were used to measure the pressure loss. Hence, the pressure loss shown in Figure 3.6 can provide a rough estimate for the expected pressure loss in experiments. As shown in Figure 3.6, at the same mass flow rate, the 2D solution always underestimates the pressure loss because of not including the side wall effects. Also, differences between the solid line and dashed line increase as the mass flow rate increases. Thus, it is easier to observe the difference between a no-slip surface and a surface with slip effects at a relatively high flow rate. Moreover, these two models can be used to determine the effective slip length, when the flow rate and pressure loss are given by simulations or experiments.

3.2.2 Computational fluid dynamics (CFD) models

The computational fluid dynamics (CFD) model was created based on Samaha’s ideas [26]. Using Samaha’s [26] assumptions of flat gas-liquid interfaces and no shear forces on the gas-liquid interface, two models were developed in this work: a full-scale model to have a better understanding of the experimental results and a unit cell model to reveal the pattern-dependent influence on the flow behavior adjacent to the surface. In addition, the simulation results of the unit cell model were used to calculate the effective slip length, which provided some insights for the surface design optimization.

3.2.3 Effective slip length estimation with computational fluid dynamics model

As addressed in Chapter 1, the analytical formulas for effective slip length estimation are only available for the four patterns illustrated in Figure 1.3. To predict the effective slip length of a given surface, a CFD-based method is proposed. This method is composed of two parts: one is a CFD model, which aims to compute the average velocity, $\bar{u}$, and pressure gradient, $\left( \frac{\partial p}{\partial x} \right)$, of the flow over the surface of interest; the other one is an analytical model that computes the effective slip length with the same average velocity and pressure gradient as the models derived in Chapter 2. If the Navier slip length, $b$, can be expressed in terms of average velocity and pressure gradient according to the analytical models, the effective slip length can be solved explicitly. If not, the effective slip length can only be solved iteratively.
Note that the analytical model can also be used to calculate the effective slip length if the average velocity and pressure gradient are measured from experiments. Figure 3.7 shows the procedure for calculating the effective slip length based on either experimental measurements or theoretical results.

3.2.4 Methods for the effective slip length calculation

For determining $b_{\text{eff}}$, the flow rate and pressure loss are measured with the same configuration for a no-slip surface (reference surface) and a surface with an effective slip length of $b_{\text{eff}}$ (test surface). Using these results, the following relations can be established:

$$
\left( \frac{\bar{u}_{\text{test}}}{\bar{u}_{\text{ref}}} \right) / \left( \frac{\Delta p_{\text{test}}}{\Delta p_{\text{ref}}} \right) = G(b_{\text{eff}}) \tag{3.2}
$$
\[ b_{\text{eff}} = F \left[ \left( \frac{\bar{u}_{\text{test}}}{\bar{u}_{\text{ref}}} \right) / \left( \frac{\Delta p_{\text{test}}}{\Delta p_{\text{ref}}} \right) \right] \]  \hspace{1cm} (3.3)
mounted on the top of the lubrication cavity (Figure 3.1) with a laser-cut double-sided tape, which is illustrated in Figure 3.1(b) and Figure 3.3. To prevent leakage, a marine adhesive sealant was used to seal the gap between the two parts. The inlet, outlet, and pressure probe ports were connected to the syringe pump, a reservoir, and differential pressure transducer (OMEGA, PX-409DIFF) via tubing, respectively.

![Figure 3.8: The flow cell design (top view and cross-section view). The dimensions of the polyimide sheet can be found in Figure 3.1.](image)

In experiments, the fluid (i.e. water) was driven by the syringe pump at different flow rates, while the corresponding pressure loss was measured by the differential pressure transducer with a resolution of 2 Pa. With the same procedure, the pressure loss was measured twice: once for a laser-cut flat sheet (i.e. no-slip reference surface); and the other for a laser-cut porous sheet (i.e. test surface). The lubrication cavity could be filled with either liquid (e.g. water) or gas (e.g. air). When the cavity fluid (lubricant) differs from the fluid (i.e. water) flowing in the channel, the lubricant might leak into the channel, which is challenging to avoid. Therefore, the lubrication cavity was filled with water for collecting preliminary data.
3.2.6 Design and construction of a system for measuring pressure loss (final)

The preliminary version of the system for measuring pressure loss had several shortcomings. First, the assembly procedure was time-consuming, because the marine adhesive took 48 hours to fully cure for a watertight seal. Second, the channel height might be inconsistent due to the selected assembly method (i.e. using a C-clamp and marine adhesive), so the test surface and the reference surface were not measured under the same conditions. Thus, the inconsistent channel height violated the condition required by the method of calculating the effective slip length (described in Section 3.2.4). Third, the pressure loss was small compared with the full measurement range of the differential pressure transducer. Such a measurement had a large relative error and relatively low accuracy because of the non-linearity of the sensor. As a result, the experimental data collected were less reliable.

To resolve the issues addressed above, an improved flow cell was designed and constructed, the exploded view of which is shown and labeled in Figure 3.9. The improved flow cell has two parts: top lid and bottom base. The two parts were fastened together by screws, and a laser-cut gasket was placed between the two parts and fitted into the slots engraved in both parts. The channel was milled with a height of 1 mm and a width of 12 mm on the underside of the top lid. The inlet, outlet, and the four ports with a spacing of 35 mm connected to gauge pressure transducers were also embedded on the top lid. The four gauge pressure transducers were denoted as Sensor A, Sensor B, Sensor C, and Sensor D based on their locations from the inlet to the outlet. The flat and perforated polyimide sheets were taped on the top of the bottom base with double-sided tape. The lubrication (gas/liquid) cavity was embedded in the bottom base and located under the perforated polyimide sheet. The lubrication (gas/liquid) cavity was connected to a syringe (not displayed in Figure 3.9) via the gas/liquid path. The improved flow cell, a custom-built syringe pump, and a reservoir (i.e. a water bottle) were connected via tubing, which formed the measurement system.

With the improved flow cell, the time used for assembling and initiating the measurement system was reduced to several hours. Because both the pressure loss over the flat sheet (reference surface) and the perforated sheet (test surface) were measured in the same assembly simultaneously, the conditions that are required by the method of calculating the effective slip length (described in Section 3.2.4) were naturally satisfied. The gauge pressure mea-
sured in the system fell into the middle of the full measurement range of the selected gauge pressure transducer (OMEGA, PX-409), which made the measurements more accurate. The gauge pressure in the measurement system can be affected by many factors, such as the water level in the reservoir, vibrations in the building, flow rate, length, dimensions, and boundaries of the water path. However, the difference between the adjacent gauge pressure transducer readouts is the pressure loss, which was not affected by the water level in the reservoir or the vibration in the building.

Figure 3.9: Exploded view of the improved flow cell. The dimensions of the perforated sheet are given in Figure 3.1, and the flat sheets are the same size as the perforated one. All of the sheets are made of polyimide.
The brief measurement procedure is similar to the measurement procedure described in Section 3.2.5, and the detailed procedure will be described with the corresponding experimental results. The lubrication cavity could be filled with either liquid phase lubrication (e.g. water) or gas phase lubrication (e.g. air). The fluid (i.e. water) was driven by the syringe pump at different flow rates, and the gauge pressures at different locations were measured by the transducers, thus the pressure loss was determined based on the gauge pressure transducer readouts. With the current measurement system configuration, the pressure loss over the flat polyimide sheet (reference surface) and perforated polyimide sheet (test surface) are measured at the same time. To avoid air leaks into the channel when the lubrication cavity was filled with air, the mass flow rate was limited below 2.764 g/s, and a soft start (slow acceleration) of the pumping was necessary.
Chapter 4
RESULTS AND DISCUSSION

In this chapter, the challenges of the topic and the significance of this work will be discussed first. The concepts that will be used in the analysis will be introduced in the following section. Then, the experimental considerations will be demonstrated with the description of the experimental procedures and uncertainty analysis. Next, the phenomenon observed in experiments, the experimental results, and the numerical results are discussed in detail. Finally, some suggestions on the surface design optimization are presented at the end of this chapter.

4.1 Challenges and significance

Researchers have made various attempts to reduce drag using surface modification because of the potential impact of such technology on energy saving and the associated economic benefits. Achieving large drag reduction and reducing the difficulty in fabrication of surface modifications have been motivating researchers and engineers for years, so many engineered surfaces were developed and tested in the laboratory, such as superhydrophobic (SHPo) surfaces or lubrication-impregnated (LI) surfaces. However, both SHPo surfaces and LI surfaces are fabricated by introducing nano-scale or micro-scale structures on the surfaces, limiting the production and application of these two kinds of surfaces. The surfaces considered in the present work do not have nano-scale or micro-scale features so they have the potential to be fabricated by high-precision tools that have been widely used in the manufacturing industry.

Drag reduction (DR) is defined in Equation (1.2), which is the most intuitive quantity that shows how much drag is reduced in a relative manner. However, DR = 1/(1 + H/b_{eff}) for Couette flow [45], and DR = 3/(3 + H/b_{eff}) for Poiseuille flow [49] show that the drag reduction not only depends on slip effects (represented by b_{eff}) on boundaries but also related
to the length scale $H$ of the flow system. As for the flow configuration discussed in this work, \( \text{DR} = 3/(4 + H/b_{\text{eff}}) \) is obtained by approximating the fluid channel as two parallel flat plates. Hence, there are two findings: (1) the drag reduction decreases as the height of the fluid channel increases even if the effective slip length at the walls does not change; and (2) the drag reduction obtained in different flow configurations cannot be directly compared with each other. For example, the effective slip length of 100 µm entails a 38% drag reduction in a 500 µm high channel, a 23% drag reduction in a 1 mm high channel, and a 3% drag reduction in a 5 mm high channel. Hence, in addition to the comparison of the drag reduction, the effective slip length is also used to represent the drag reduction performance of a given surface.

The effective slip length $b_{\text{eff}}$ has been widely used as a measure of the drag reduction effect defined in Equation (1.3). However, the effective slip length can also be influenced by other quantities, such as shear rate $\dot{\gamma}$ and Re. It is worth mentioning that the same author might use different models to fit the experimental data and claim the applicability of each fitting method. For example, both $b_{\text{eff}} = A + B\dot{\gamma}$ [50] and $b_{\text{eff}} = A \cdot \dot{\gamma}^H$ [51] are used by Choi to analyze the effects of the shear rate on the effective slip length $b_{\text{eff}}$, which seems likely due to the spread of the data and limited test ranges. In Cheng’s theoretical works [52], the effects of Reynolds number on the effective slip length was demonstrated, and such dependence could be altered by the porosity and the feature size.

Up to this point, the non-flat slip surface has not been discussed yet, the effect of which cannot be ignored when the feature size (e.g. the bubble height) is comparable to the length scale of the flow system (e.g. the channel height is the typical length scale for the flow in a channel). In 2007, Steinberger [22] demonstrated the evolution of the effective slip length with the meniscus shape characterized by the protrusion angle $\theta$ for an array of micro-scale holes on a rectangular lattice, where the decrease of $b_{\text{eff}}$ for $\theta > 45^\circ$ was shown. Then, Carlborg [20] developed a device that actively controlled drag reduction via controlling the shape of the air-water interface, and a similar device was used by Karatay [4] to investigate the effect of the air-water interface shape (characterized by $\theta$) on the effective slip length. Karatay [4] confirmed Steinberger’s findings [22] in a different experimental configuration with micron resolution particle image velocimetry ($\mu$PIV). In this work, the slippage was
introduced by air bubbles trapped in the perforated sheet, and the hole size was comparable to the channel height, so the effective slip lengths obtained from experiments were affected by the shape of the air-water interface.

Several experimental challenges must be faced when the slip surface is introduced by a gas-liquid (e.g. air-water) interface. First, the air-water interface is vulnerable to rupture when there is a large pressure change across the interface. Without a real-time air pressure compensation system (which is the current situation), disturbances to the air-water interface have to be passively resisted by the surface tension, which highly depends on the size and the shape of the structure in which the air pocket is trapped. Emami et al. [47,48] developed a numerical model to predict the shape and stability of the air-water interface. According to Emami’s results, the air-water interface trapped in the circular hole has the highest critical pressure so the circular hole is the desired shape for robustness compared with other common shapes, such as triangles, hexagons, and ellipses.

To estimate the critical pressure for rupture of the air-water interface trapped in a circular hole, the Laplace pressure model can be used. The Laplace pressure is given by

\[ \Delta P_{\text{Laplace}} = P_{\text{inside}} - P_{\text{outside}} = \frac{2\sigma}{R} \]  \hspace{1cm} (4.1)

where \( \Delta P_{\text{Laplace}} \), \( P_{\text{inside}} \), \( P_{\text{outside}} \), \( \sigma \), and \( R \) represent the pressure difference across the air-water interface, the pressure inside the air bubble, the pressure outside the air bubble, the surface tension, and the radius of curvature, respectively. \( R \) can be determined by the hole diameter and the protrusion angle of the air bubble \( \theta \) (defined by the angle between the tangent of the air-water interface and the diameter of the hole) i.e. \( R = (d/2)/\sin \theta \). Assuming \(-90^\circ < \theta < 90^\circ\) and substituting \( \sigma = 72.86 \times 10^{-3} \text{ N/m} \) and the corresponding hole diameter \( d \), the Laplace pressure can be plotted as a function of the protrusion angle \( \theta \), shown in Figure 4.1. Due to the compressibility and the thermodynamic properties of air, the pressure in the air-side will change as the volume changes due to the displacement of the air-water interface. For simplicity, the thermodynamic process is assumed isothermal, and the air is considered as an ideal gas.
From the ideal gas law,

\[ pV = nR_uT \]  \hspace{1cm} (4.2)

where \( p \) is the absolute pressure, \( V \) is the volume, \( n \) is the number of moles of the gas, \( R_u \) is the universal gas constant, and \( T \) is the absolute temperature. For the isothermal process, the volume and pressure of a system at the final state can be found based on the volume and pressure of the same system at the initial state as follows

\[ p_i V_i = p_f V_f \]  \hspace{1cm} (4.3)

where \( V \) and \( p \) represents the volume and pressure of the system, respectively. The subscript
‘i’ and ‘f’ denotes the initial state and final state, correspondingly. Denoting the volume change due to the displacement of the air-water interface as $\Delta V$ and expressing $V_f$ in terms of $V_i$ and $\Delta V$ as $V_f = V_i + \Delta V$, the pressure change due to the volume change ($\Delta P_{\text{volume}}$) can be obtained from Equation (4.3). Hence, a better estimation of the pressure that the air-water interface is able to sustain can be given by

$$\Delta P = \Delta P_{\text{volume}} + (\Delta P_{\text{Laplace}})$$

(4.4)

where $(-\Delta P_{\text{Laplace}})$ represents the pressure that are held by the surface tension of the air-water interface. Equation (4.4) was used to assist the surface design to keep the trapped air bubble stable, which is one of the experimental challenges. Equation (4.4) also provides an estimate needed for optimizing the soft start of the syringe pump described earlier. Using the method proposed by Gerlach et al. [53], a better estimate of the pressure difference across the air-water interface can be obtained, which can be used to replace $\Delta P_{\text{Laplace}}$ to provide a more accurate estimate of $\Delta P$. If the air bubbles escape from where they are trapped, it will significantly interfere with the pressure loss measurements. Bubble escape was avoided by improving the surface design and using a soft start of the syringe pump.

Another challenge to measuring drag reduction is measuring pressure loss in the experiments because the pressure loss over the perforated sheet (test surface) and the pressure loss over the flat sheet (reference surface) differed by less than 100 Pa, which is equivalent to the hydrostatic pressure generated by a 10 mm-high water column. Such a small difference in pressure loss requires highly accurate pressure measurements, which are naturally vulnerable to ambient noise.

In this work, drag reduction values greater than 20% and the effective slip lengths greater than 90 $\mu$m were achieved in experiments. These achievements are significant although caution needs to be taken when comparing the drag reduction and the effective slip length with the previously published results.

### 4.2 Characteristic parameters

For the convenience of the following discussion and comparing the current work with published results, several characteristic parameters that have been widely used by researchers
are employed and summarized in this section. The Reynolds number, the most important
dimensionless parameter to specify a fluid flow in a rectangular duct, is defined by

\[ \text{Re} = \frac{\bar{u}D_h}{\nu} \]  

(4.5)

where \( \bar{u} \) is the average flow velocity, \( \nu \) is the kinematic viscosity of the fluid, and \( D_h \) is the
hydraulic diameter given by

\[ D_h = \frac{4A_c}{\Pi} \]  

(4.6)

where \( A_c \) is the cross sectional area of the rectangular duct, and the \( \Pi \) is its perimeter.

The drag coefficient \( C_D \) describes the drag of the immersed body in dimensionless form,
and its definition is given by

\[ C_D = \frac{F_D}{\frac{1}{2} \rho U^2 A} \]  

(4.7)

where \( \rho \) is the density of the fluid, \( U \) is the velocity scale, \( A \) is a reference area and \( F_D \) is
the drag force, respectively.

For the flows involving surface tension \( \sigma \), the Capillary number, \( \text{Ca} \), and Weber number,
\( \text{We} \), are commonly used

\[ \text{We} = \frac{\rho U^2 l}{\sigma} \]  

(4.8)

\[ \text{Ca} = \frac{\mu U}{\sigma} \]  

(4.9)

where \( l \) is the length scale, and \( \mu \) is the dynamic viscosity. At high Weber number, droplets
and bubbles can be deformed easily as the fluid accelerates or decelerates. At high capillary
number, the viscous forces dominate those from surface tension.

In this work, the average velocity \( \bar{u} \) is used as the velocity scale \( U \), and the height of the
channel \( H \) is used as the length scale \( l \).
4.3 Experimental procedures

The experimental apparatus described in detail in Section 3.2.6 is used to measure the pressure loss over the flat sheets (reference surfaces) and the perforated sheets (test surfaces) at different mass flow rates.

The different mass flow rates (from 1.382 g/s to 2.764 g/s) were produced by an in-house syringe pump, which was controlled by the lab PC. The custom syringe pump consisted of a linear translation stage, a stepper motor and its driver, a syringe, and a controller (Arduino Uno). With the stepper motor speeds (expressed in the unit of pulse/s), the dimensions, and the specifications of all the components, the expected mass flow rate could be calculated. In addition, a laser displacement sensor (optoNCDT 1420) was used to monitor the performance of the linear stage that was driven by the stepper motor. Based on the readout from the displacement sensor (optoNCDT 1420) and the diameter of the syringe, the measured mass flow rate was obtained. For all the stepper motor speeds used in collecting data, the corresponding expected mass flow rates and measured mass flow rates are listed in Table 4.1, which indicates the custom syringe pump operated as expected.

<table>
<thead>
<tr>
<th>Stepper motor speed</th>
<th>Mass flow rate (expected)</th>
<th>Mass flow rate (measured)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1280 pulse/s</td>
<td>1.382 g/s</td>
<td>1.382 g/s</td>
</tr>
<tr>
<td>1600 pulse/s</td>
<td>1.727 g/s</td>
<td>1.711 g/s</td>
</tr>
<tr>
<td>1920 pulse/s</td>
<td>2.073 g/s</td>
<td>2.069 g/s</td>
</tr>
<tr>
<td>2240 pulse/s</td>
<td>2.418 g/s</td>
<td>2.400 g/s</td>
</tr>
<tr>
<td>2560 pulse/s</td>
<td>2.764 g/s</td>
<td>2.746 g/s</td>
</tr>
</tbody>
</table>

At each mass flow rate listed in Table 4.1, the gauge pressure was measured simultaneously by the four gauge pressure transducers, and these gauge pressure readouts were used to compute the pressure loss over the flat sheets (reference surfaces) and the perforated sheets.
(test surfaces) according to the sensor locations. As shown in Figure 3.9, the four gauge pressure transducers are denoted as Sensor A, Sensor B, Sensor C, and Sensor D based on their locations from the inlet to the outlet. The raw data collected from each gauge pressure transducer can be plotted as a function of time as shown in Figure 4.2 and Figure 4.3. The data presented in Figure 4.2 and Figure 4.2 were taken at the same mass flow rate i.e. 2.764 g/s but were collected with the lubrication cavity filled with water and with the lubrication cavity filled with air, respectively.

Figure 4.2: Gauge pressure as a function of time when the lubrication cavity was filled with water. $P_A$, $P_B$, $P_C$, and $P_D$ represent the gauge pressure values that were measured by Sensor A, Sensor B, Sensor C, and Sensor D, respectively.
The pressure profiles shown in Figure 4.2 and Figure 4.3 can be divided into three regions according to the inflection points: before-pumping, pumping, and after-pumping. Note that the gauge pressure in the after-pumping region is higher than that in the before-pumping region and the gauge pressure keeps increasing in the pumping region. The difference and the increment shown in the gauge pressure profiles are due to the water level change in the reservoir.

Figure 4.3: Gauge pressure as a function of time when the lubrication cavity was filled with air. $P_A$, $P_B$, $P_C$, and $P_D$ represent the gauge pressure values that were measured by Sensor A, Sensor B, Sensor C, and Sensor D, respectively.
When the lubrication cavity was filled with air, the air pockets were trapped in the holes of the perforated sheet. The air-water interface is vulnerable to the disturbances, such as pressure impulse generated at the syringe pump start-up phase. If the stability of the air-water interface is broken, the air can escape from the cavity to the channel, which will interfere with the pressure loss measurements. Hence, a soft start of the syringe pump was necessary when the lubrication cavity was filled with air. The soft-start was achieved by setting a smaller acceleration for the stepper motor, the effects of which can be visualized easily by the slope of the gauge pressure profile in the transition between the before-pumping region and pumping region, in Figure 4.2 and Figure 4.3.

The pressure losses over the flat sheet (reference surface) and the perforated sheet (test surface) were computed from the gauge pressure measurements by

\[ \Delta p_{ref} = (\bar{p}_{A, \text{pumping}} - \bar{p}_{A, \text{before-pumping}}) - (\bar{p}_{B, \text{pumping}} - \bar{p}_{B, \text{before-pumping}}) \]  

\[ \Delta p_{test} = (\bar{p}_{C, \text{pumping}} - \bar{p}_{C, \text{before-pumping}}) - (\bar{p}_{D, \text{pumping}} - \bar{p}_{D, \text{before-pumping}}) \]  

where the over bar ‘\( \bar{\cdot} \)’ represents the mean value. Because the mean of the sum of two variables equals the sum of their means, Equation 4.10 and Equation 4.11 can be rewritten in the following form

\[ \Delta p_{ref} = (p_{A, \text{pumping}} - p_{B, \text{pumping}}) - (p_{A, \text{before-pumping}} - p_{B, \text{before-pumping}}) \]  

\[ \Delta p_{test} = (p_{C, \text{pumping}} - p_{D, \text{pumping}}) - (p_{C, \text{before-pumping}} - p_{D, \text{before-pumping}}) \]  

Note that the gauge pressures were measured simultaneously and the collected data were aligned in time. If the ambient noise was present, it was captured by all the sensors and logged into the data. By doing the arithmetic operations with the data aligned in time, the influence of the ambient noise can be significantly reduced. Hence, changing the order of averaging and arithmetic operations can suppress the ambient noise in the data processing.

Utilizing Equation (4.12) and Equation (4.13), \( \Delta p_{ref} \) and \( \Delta p_{test} \) can be calculated based on the gauge pressure transducer readouts \( (p_A, p_B, p_C, \text{ and } p_D) \). To solve for the effective slip length \( b_{eff} \), \( \Delta p_{ref} \) and \( \Delta p_{test} \) were substituted into either the left hand side of Equation
or the right hand side of Equation (3.3). The right hand side of Equation (3.4) (i.e. $G$) was determined by 2D or 3D models developed in Chapter 2, and $b_{\text{eff}}$ needed to be solved iteratively. The right hand side of Equation (3.3) (i.e. $F$) could be only obtained with 2D models developed in Chapter 2, and $b_{\text{eff}}$ could be determined explicitly. Due to the simplicity of the 2D model, the expressions of the function $G$ and the function $F$ will be presented in Section 4.4.

If the effective slip length $b_{\text{eff}}$ is known, the peak velocity $u_{\text{max}}$ and the shear rate $\dot{\gamma}$ at the perforated surface can be computed by Equation (2.20) and Equation (2.21) with the 3D assumption, and by Equation (2.26) and Equation (2.27) with the 2D approximation.

The effective slip velocity $u_{\text{slip, eff}}$ is defined by

$$u_{\text{slip, eff}} = b_{\text{eff}} \left( \frac{\partial u}{\partial y} \right) \bigg|_{\text{at the surface}} = b_{\text{eff}} \cdot \dot{\gamma}$$

Equation (4.14) is used to calculate $u_{\text{slip, eff}}$ with $b_{\text{eff}}$ and $\dot{\gamma}$. Also, the effective slip velocity $u_{\text{slip, eff}}$ can be used as a boundary condition for a boundary layer model (external flow) to estimate the drag reduction effects, which extends the experimental findings obtained from the flow cell (internal flow) to a boundary layer problem (external flow). A brief discussion of the drag reduction effects in the external flow can be found in Chapter 5.

To install or replace the perforated sheet, the flow cell had to be reassembled, which might cause channel height changes. Although the height changes are small (not noticeable by human eyes), they could significantly alter the pressure loss due to the dimensions of the flow cell. As a consequence, the pressure loss measured over different perforated sheets cannot be directly compared with each other. Also, the channel height is needed to compute the effective slip length $b_{\text{eff}}$ so the change of the height of the channel has to be known to get a more accurate $b_{\text{eff}}$. Thus, Kashaninejad’s idea [54] was adapted to this work, that is, the pressure loss over the flat film was used to calibrate the height of the channel with the 3D analytical model. The maximum difference between the calibrated channel height and the nominal channel height (800 µm) was 57 µm.
The procedures described above were used to investigate different perforated films. The pressure losses presented or used in the calculation are the average values of three experiments at the same mass flow rate and with a lubrication cavity filled with the same fluid.

4.4 Uncertainty analysis

As described in Section 3.2.4, the effective slip length has the following relation:

\[
\frac{\bar{u}_{\text{test}}}{\bar{u}_{\text{ref}}} / \left( \frac{\Delta p_{\text{test}}}{\Delta p_{\text{ref}}} \right) = G(b_{\text{eff}}) \tag{3.2}
\]

\[
b_{\text{eff}} = F \left[ \frac{\bar{u}_{\text{test}}}{\bar{u}_{\text{ref}}} / \left( \frac{\Delta p_{\text{test}}}{\Delta p_{\text{ref}}} \right) \right] \tag{3.3}
\]

where the function \( F \) can be explicitly given only if the flow was between two parallel plates as shown in Figure 4.4. Hence, this configuration is used to estimate the uncertainty of the experimental measurements, although the data was collected in the rectangular channel described in Section 3.2.6.

Figure 4.4: Schematic view of two parallel plates with different slip lengths \( (b_+, b_-) \) at both top and bottom walls, where \( H \) represents the distance between two parallel plates.
Based on the analytical model (2D) developed by Kashaninejad [38], Equation (3.2) can be expressed in terms of the slip lengths \((b_+, b_-)\) and the height of channel, \(H\)

\[
\left[ \frac{\bar{u}_{\text{test}}}{\bar{u}_{\text{ref}}} \right] / \left[ \frac{\Delta p_{\text{test}}}{\Delta p_{\text{ref}}} \right] = \frac{H + 4b_+ + 4b_- + 12b_+ b_-}{H + b_+ + b_-} \quad (4.15)
\]

Let \(b_- = b_{\text{eff}}, b_+ = 0, \Theta = \bar{u}_{\text{test}}/\bar{u}_{\text{ref}},\) and \(\Gamma = \Delta p_{\text{test}}/\Delta p_{\text{ref}}.\) Equation (4.15), can be simplified and rewritten as

\[
\Theta/\Gamma = \frac{H + 4b_{\text{eff}}}{H + b_{\text{eff}}} \quad (4.16)
\]

which also gives \(G(b_{\text{eff}}) = (H + 4b_{\text{eff}})/(H + b_{\text{eff}}).\) Also, \(\text{DR} = 3/(4 + H/b_{\text{eff}})\) can be deduced from Equation (4.16). Rearranging Equation (4.16), the effective slip length \(b_{\text{eff}}\) as well as \(F\) can be expressed as

\[
b_{\text{eff}} = F(\Theta, \Gamma, H) = \frac{\Theta/\Gamma - 1}{4 - \Theta/\Gamma} \cdot H = \frac{\Theta - \Gamma}{4\Gamma - \Theta} \cdot H \quad (4.17)
\]

Thus, the uncertainty of \(b_{\text{eff}}\) can be given by

\[
\delta b_{\text{eff}} = \sqrt{\left[ \left( \frac{\partial F}{\partial \Theta} \right) \delta \Theta \right]^2 + \left[ \left( \frac{\partial F}{\partial \Gamma} \right) \delta \Gamma \right]^2 + \left[ \left( \frac{\partial F}{\partial H} \right) \delta H \right]^2} \quad (4.18)
\]

where

\[
\left( \frac{\partial F}{\partial \Theta} \right) = \frac{H}{4\Gamma - \Theta} + \frac{(\Theta - \Gamma)H}{(4\Gamma - \Theta)^2} \quad (4.19a)
\]

\[
\left( \frac{\partial F}{\partial \Gamma} \right) = -\frac{H}{4\Gamma - \Theta} - \frac{4(\Theta - \Gamma)H}{(4\Gamma - \Theta)^2} \quad (4.19b)
\]

\[
\left( \frac{\partial F}{\partial H} \right) = \frac{\Theta - \Gamma}{4\Gamma - \Theta} \quad (4.19c)
\]
Because \( \left( \frac{\partial F}{\partial \Theta} \right) \propto H \) and \( \left( \frac{\partial F}{\partial \Gamma} \right) \propto H \), increasing the height of the channel will lead to higher uncertainty in the measurement results, which should be avoided in experiments. The upper limit of the uncertainty can be estimated based on the experimental conditions and measurements i.e. \( \Theta = 1, \delta \Theta = 0, \Gamma = 0.65, \delta \Gamma = 0.05, H = 0.800 \text{ mm}, \delta H = 0.060 \text{ mm} \). The estimated uncertainty of the effective slip length is \( \delta b_{\text{eff}} = 31.554 \mu\text{m} \). Therefore, all the experimental results have enough significance to prove that the surface gas/liquid entrapment can reduce drag.

4.5 Experimental results

4.5.1 Overview

Based on the ideas described in Chapter 3 and previous sections of this chapter, circular holes on a hexagonal lattice with a horizontal row (aligned with the flow direction) are selected as the hole shape and layout, which are shown in Figure 3.2. Such a design can be described without ambiguity by two diameters, i.e., the hole diameter, \( d \), and the center-to-center spacing between holes, \( s \). The designs were fabricated on the polyimide sheets using the laser micromachining system described in Section 3.1.2 and following the procedures described in Section 4.3. These processed sheets, called the perforated sheets (test surface), were taped on the top of the lubrication cavity with a layer of double-sided tape cut to the required shape, the exploded view of which is shown in Figure 3.4. The lubrication cavity could be filled with either water or air producing different experimental conditions.

Table 4.2 summarizes the cases that were experimentally investigated with the system described in Section 3.2.6. Specifically, the key parameters (the hole diameter and the center-to-center spacing between holes) are listed, and the corresponding local porosity, \( \phi_{\text{local}} \), the cavity porosity, \( \phi_{\text{cavity}} \), and the test porosity, \( \phi_{\text{test}} \), are also calculated and provided in Table 4.2. The last column of Table 4.2 states the experimental conditions via specifying the medium used in the lubrication cavity.
Table 4.2: Summary of the cases experimentally investigated.

<table>
<thead>
<tr>
<th>Case No.</th>
<th>d (mm)</th>
<th>s (mm)</th>
<th>$\phi_{\text{local}}$</th>
<th>$\phi_{\text{cavity}}$</th>
<th>$\phi_{\text{test}}$</th>
<th>Cavity fluid</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1.0</td>
<td>1.2</td>
<td>62.98%</td>
<td>56.10%</td>
<td>37.40%</td>
<td>water</td>
</tr>
<tr>
<td>2</td>
<td>1.0</td>
<td>1.5</td>
<td>40.31%</td>
<td>31.70%</td>
<td>21.13%</td>
<td>water</td>
</tr>
<tr>
<td>3</td>
<td>1.0</td>
<td>1.2</td>
<td>62.98%</td>
<td>56.10%</td>
<td>37.40%</td>
<td>air</td>
</tr>
<tr>
<td>4</td>
<td>0.8</td>
<td>1.2</td>
<td>40.31%</td>
<td>35.90%</td>
<td>23.94%</td>
<td>air</td>
</tr>
<tr>
<td>5</td>
<td>0.6</td>
<td>1.2</td>
<td>22.67%</td>
<td>20.20%</td>
<td>13.46%</td>
<td>air</td>
</tr>
</tbody>
</table>

4.5.2 Lubrication cavity filled with water

For tests with the lubrication cavity filled with water, the system described in Section 3.2.6 was filled with water to eliminate any unwanted air bubbles. As a consequence, when the system was already primed, the lubrication cavity was filled with water and was ready for pressure loss measurements. This configuration is called a water-filled case.

Two water-filled cases i.e. Case 1 and Case 2 were experimentally investigated, and the results are summarized in Table 4.3 and Table 4.4, respectively. For each mass flow rate, $\dot{m}$, the pressure loss over the flat sheet, $\Delta p_{\text{ref}}$, and the pressure loss over the perforated sheet, $\Delta p_{\text{test}}$, were measured experimentally, to determine the drag reduction. The tabulated values of $\Delta p_{\text{ref}}$ and $\Delta p_{\text{test}}$ are the averages from three experiments. Using Equation (3-48) in White’s book [55], $\Delta p_{\text{ref}}$ was used to compute the corrected channel height $H_{\text{cor}}$, which is needed to calculate the Reynolds number.
Table 4.3: Re, $H_{\text{cor}}$, $\Delta p_{\text{ref}}$, $\Delta p_{\text{test}}$ and DR as a function of $\dot{m}$ for Case 1.

<table>
<thead>
<tr>
<th>$\dot{m}$ (g/s)</th>
<th>Re</th>
<th>$H_{\text{cor}}$ (mm)</th>
<th>$\Delta p_{\text{ref}}$ (Pa)</th>
<th>$\Delta p_{\text{test}}$ (Pa)</th>
<th>DR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.382</td>
<td>213</td>
<td>0.846</td>
<td>84</td>
<td>28</td>
<td>66.6%</td>
</tr>
<tr>
<td>1.727</td>
<td>266</td>
<td>0.845</td>
<td>106</td>
<td>40</td>
<td>62.0%</td>
</tr>
<tr>
<td>2.073</td>
<td>320</td>
<td>0.838</td>
<td>130</td>
<td>53</td>
<td>59.0%</td>
</tr>
<tr>
<td>2.418</td>
<td>373</td>
<td>0.831</td>
<td>155</td>
<td>67</td>
<td>57.0%</td>
</tr>
<tr>
<td>2.764</td>
<td>427</td>
<td>0.823</td>
<td>183</td>
<td>81</td>
<td>56.0%</td>
</tr>
</tbody>
</table>

In Case 1, a large drag reduction of up to 66.6% was observed at the lowest mass flow rate (1.382 g/s), and the drag reduction decreased down to 56.0% as the mass flow rate increased to its maximum (2.764 g/s). Similar results and trends were also observed in Case 2, that is, the drag reduction decreased from 49.3% to 25.2% as the mass flow rate increased from

Table 4.4: Re, $H_{\text{cor}}$, $\Delta p_{\text{ref}}$, $\Delta p_{\text{test}}$ and DR as a function of $\dot{m}$ for Case 2.

<table>
<thead>
<tr>
<th>$\dot{m}$ (g/s)</th>
<th>Re</th>
<th>$H_{\text{cor}}$ (mm)</th>
<th>$\Delta p_{\text{ref}}$ (Pa)</th>
<th>$\Delta p_{\text{test}}$ (Pa)</th>
<th>DR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.382</td>
<td>213</td>
<td>0.860</td>
<td>81</td>
<td>41</td>
<td>49.3%</td>
</tr>
<tr>
<td>1.727</td>
<td>266</td>
<td>0.859</td>
<td>101</td>
<td>59</td>
<td>41.9%</td>
</tr>
<tr>
<td>2.073</td>
<td>319</td>
<td>0.859</td>
<td>121</td>
<td>78</td>
<td>36.1%</td>
</tr>
<tr>
<td>2.418</td>
<td>373</td>
<td>0.861</td>
<td>140</td>
<td>97</td>
<td>30.7%</td>
</tr>
<tr>
<td>2.764</td>
<td>426</td>
<td>0.857</td>
<td>163</td>
<td>122</td>
<td>25.2%</td>
</tr>
</tbody>
</table>
1.382 g/s to 2.764 g/s. Such large drag reduction resulted not only from replacing liquid-solid boundaries with liquid-liquid boundaries but also from the water bypassing through the lubrication cavity, which was confirmed in both experiments and simulations. In experiments, dye (food coloring) was added to the lubrication cavity after the measuring system had been primed. When the flow was driven by the syringe pump, the dye was carried into the main channel, which demonstrated the presence of bypass flow. Unfortunately, the contribution of the bypass effect on the drag reduction was not investigated experimentally, due to the lack of the proper diagnostic methods. However, the bypass effect on the pressure distribution along the flow direction was studied using a computational fluid dynamics model, the results of which will be presented in Section 4.7.1.

The drag reduction is plotted as a function of the mass flow rate for Case 1 and Case 2 in Figure 4.5. At the same flow rate, the perforated sheet used in Case 1 produces more drag reduction compared with the one used in Case 2, which is due to two reasons: (1) the perforated sheet in Case 1 has a larger area of liquid-liquid interface (i.e. slip surface) compared with Case 2, because of the different hole spacing ($s = 1.2$ mm in Case 1 and $s = 1.5$ mm in Case 2); and (2) for the same reason, the resistance for the bypass flow from the holes is smaller in Case 1 compared with Case 2 because, in Case 1, more holes allow the bypass flow to pass through. In addition, the decrease in drag reduction with increasing the mass flow rate might be caused by the resistance experienced by the bypass flow, which increases as the mass flow rate increases. The resistance experienced by the bypass flow includes the resistance due to the pass-through effect of the holes and the entrance and exit effects of the lubrication cavity, all of which increase as the mass flow rate increases.
Figure 4.5: The drag reduction DR as a function of the mass flow rate $\dot{m}$ for Case 1 ($d = 1.0 \text{ mm}, s = 1.2 \text{ mm}$) and Case 2 ($d = 1.0 \text{ mm}, s = 1.5 \text{ mm}$).

4.5.3 Lubrication cavity filled with air

When the lubrication cavity needs to be filled with air, the test procedure becomes more challenging. Following the procedures described in Section 3.2.6 for the initialization of the measuring system, the whole flow cell including the lubrication cavity would be filled with water i.e. no air bubbles. Then, air was injected into the lubrication cavity by a syringe via the lubrication path marked in Figure 3.1. The air injection should be slow and performed carefully to prevent the air from going into the main channel of the flow cell. When the air bubbles enter the main channel, they were randomly trapped somewhere in the flow cell. If the air bubbles become trapped in the ports connected to the pressure transducer, the
readouts of the sensors would no longer reflect the correct pressure loss, so re-initialization would be required to remove the unwanted air bubbles.

Figure 4.6: Schematic illustration of the air-water interface of entrapped air pockets.

Depending on the volume of air added to the lubrication cavity, the air-water interface could be convex, concave, or flat depending on the balance of air pressure, water pressure, and surface tension, as shown in Figure 4.6. The effect of the trapped air bubbles on drag reduction in a channel has been discussed by Gatapova et al. [56]. In experiments, the air-water interface preferred a convex shape after the air injection and could stay stable for hours because the hole size used in Case 3, Case 4, and Case 5, allowed the air-water interface to produce enough surface tension to resist small disturbances. Therefore, the perforated sheets that were designed and fabricated were capable of achieving robust surface air entrapment under hydrostatic conditions.

Under the hydrodynamic conditions, keeping the air-water interface stable was challenging for multiple reasons, such as fluid flow instability, the compressibility of air, and environmental noise. However, by using the syringe pump with a soft start, the escape of
air from the perforated surface has been successfully avoided.

Three air-filled cases i.e. Case 3, Case 4, and Case 5 were experimentally investigated, and the results are summarized in Table 4.5, Table 4.6, and Table 4.7, respectively. For each mass flow rate $\dot{m}$, the pressure loss over the flat sheet, $\Delta p_{\text{ref}}$, and the pressure loss over the perforated sheet, $\Delta p_{\text{test}}$, were measured experimentally to determine the drag reduction. The tabulated values of $\Delta p_{\text{ref}}$ and $\Delta p_{\text{test}}$ are the averages from three experiments. Using Equation (3-48) in White’s book [55], $\Delta p_{\text{ref}}$ was used to compute the corrected channel height, $H_{\text{cor}}$, which is needed to calculate the Reynolds number.

Table 4.5: Re, $H_{\text{cor}}$, $\Delta p_{\text{ref}}$, $\Delta p_{\text{test}}$ and DR as a function of $\dot{m}$ for Case 3.

<table>
<thead>
<tr>
<th>$\dot{m}$ (g/s)</th>
<th>Re</th>
<th>$H_{\text{cor}}$ (mm)</th>
<th>$\Delta p_{\text{ref}}$ (Pa)</th>
<th>$\Delta p_{\text{test}}$ (Pa)</th>
<th>DR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.382</td>
<td>214</td>
<td>0.807 mm</td>
<td>97</td>
<td>65</td>
<td>32.1%</td>
</tr>
<tr>
<td>1.727</td>
<td>267</td>
<td>0.806 mm</td>
<td>122</td>
<td>80</td>
<td>34.4%</td>
</tr>
<tr>
<td>2.073</td>
<td>321</td>
<td>0.807 mm</td>
<td>146</td>
<td>97</td>
<td>33.5%</td>
</tr>
<tr>
<td>2.418</td>
<td>374</td>
<td>0.807 mm</td>
<td>170</td>
<td>113</td>
<td>33.7%</td>
</tr>
<tr>
<td>2.764</td>
<td>428</td>
<td>0.801 mm</td>
<td>198</td>
<td>133</td>
<td>34.4%</td>
</tr>
</tbody>
</table>
Table 4.6: Re, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 4.

<table>
<thead>
<tr>
<th>$\dot{m}$ (g/s)</th>
<th>Re</th>
<th>$H_{cor}$ (mm)</th>
<th>$\Delta p_{ref}$ (Pa)</th>
<th>$\Delta p_{test}$ (Pa)</th>
<th>DR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.382</td>
<td>213</td>
<td>0.837 mm</td>
<td>87</td>
<td>62</td>
<td>29.0%</td>
</tr>
<tr>
<td>1.727</td>
<td>267</td>
<td>0.838 mm</td>
<td>109</td>
<td>79</td>
<td>26.8%</td>
</tr>
<tr>
<td>2.073</td>
<td>320</td>
<td>0.836 mm</td>
<td>131</td>
<td>94</td>
<td>28.2%</td>
</tr>
<tr>
<td>2.418</td>
<td>373</td>
<td>0.838 mm</td>
<td>152</td>
<td>112</td>
<td>25.5%</td>
</tr>
<tr>
<td>2.764</td>
<td>426</td>
<td>0.858 mm</td>
<td>162</td>
<td>128</td>
<td>22.7%</td>
</tr>
</tbody>
</table>

Table 4.7: Re, $H_{cor}$, $\Delta p_{ref}$, $\Delta p_{test}$ and DR as a function of $\dot{m}$ for Case 5.

<table>
<thead>
<tr>
<th>$\dot{m}$ (g/s)</th>
<th>Re</th>
<th>$H_{cor}$ (mm)</th>
<th>$\Delta p_{ref}$ (Pa)</th>
<th>$\Delta p_{test}$ (Pa)</th>
<th>DR</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.382</td>
<td>213</td>
<td>0.832 mm</td>
<td>89</td>
<td>64</td>
<td>27.6%</td>
</tr>
<tr>
<td>1.727</td>
<td>267</td>
<td>0.834 mm</td>
<td>110</td>
<td>80</td>
<td>27.9%</td>
</tr>
<tr>
<td>2.073</td>
<td>320</td>
<td>0.834 mm</td>
<td>132</td>
<td>95</td>
<td>27.6%</td>
</tr>
<tr>
<td>2.418</td>
<td>373</td>
<td>0.835 mm</td>
<td>154</td>
<td>110</td>
<td>28.6%</td>
</tr>
<tr>
<td>2.764</td>
<td>427</td>
<td>0.828 mm</td>
<td>180</td>
<td>124</td>
<td>30.8%</td>
</tr>
</tbody>
</table>

Significant drag reduction (DR $> 20\%$) was also achieved in all of the air-filled cases. A drag reduction of around 34% was observed in Cased 3 within the mass flow rate range of $1.382 \text{ g/s} \sim 2.764 \text{ g/s}$, which is the highest drag reduction achieved among the three air-filled cases. Also, the drag reduction measured in Case 4 has a greater spread ($\approx 7\%$)
compared with that in Case 3 (≈ 2%) and Case 5 (≈ 3%). Considering the limitations of the experimental method and the measurement system, drag reductions are not significantly different from each other within each case. Hence, the following conclusion can be drawn: the drag reduction seems less affected by the mass flow rate within the test range, unlike the significant negative correlation observed in water-filled cases.

To study the effect of different patterns on drag reduction, the drag reduction listed in Table 4.5, Table 4.6, and Table 4.7 are plotted in Figure 4.7, and compared with each other. The perforated sheet used in Case 3 outperforms the perforated sheets used in the other two cases, which is expected for the following reasons: (1) the perforated sheet used in Case 3 has the largest area of the slip surface; (2) the perforated sheet used in Case 3 has the largest hole size so that the air-water interface would protrude less compared with the other two cases; (3) during the measurement, the disturbances of the flow field due to the irregular vibration of the air-water interface as observed in video recordings was small. The latter (i.e. the irregular vibration) can alter the effective channel height and increase kinetic energy loss.

According to Table 4.2, the perforated sheets used in the air-filled cases have the same layout (i.e. a hexagonal lattice with center-to-center spacing between holes of $s = 1.2$ mm), but different hole diameter of $d = 1.0$ mm, $d = 0.8$ mm, and $d = 0.6$ mm. Thus, the area ratio of the holes on the perforated sheets used in the experiments is $1 : 0.64 : 0.36$ compared to Case 3.

In Case 4 and Case 5, the drag reductions do not significantly differ from each other in the measurement range of $1.382 \text{ g/s} \sim 2.073 \text{ g/s}$ and do not appear to depend significantly on the mass flow rate within the same range. However, the perforated sheet used in Case 4 has a larger slip-surface area than that of the perforated sheet used in Case 5, which predicts a higher drag reduction in Case 4. The contradiction between the experimental results and the intuitive judgment can be explained by the presence of the additional flow resistance due to the disturbances generated by the vibration of the air-water interfaces, and such vibration was observed and recorded on video during the measurement. The phenomenon that the vibration on the boundary increases the flow resistance was also found and discussed by Zhang et al. [57] in their recent numerical study.
Also, it has been experimentally observed that the vibration of the air-water interface intensifies at a high mass flow rate. As shown in Figure 4.7, the drag reduction decreases at the high mass flow rate in Case 4. Note that the disturbances that generate the additional flow resistance depend on not only the magnitude of the vibration but also the area affected by the vibration. Because of the small area of the air-water interface in Case 5, the disturbance due to the vibration of the air-water interface is limited, and the increase in flow resistance is small. Therefore, the decrease in drag reduction was not observed in Case 5.

Figure 4.7: The drag reduction DR as a function of the mass flow rate $\dot{m}$ for Case 3 ($d = 1.0 \text{ mm, } s = 1.2 \text{ mm}$), Case 4 ($d = 0.8 \text{ mm, } s = 1.2 \text{ mm}$) and Case 5 ($d = 0.6 \text{ mm, } s = 1.5 \text{ mm}$).
As discussed in Section 4.1, the drag reduction ratio highly depends on the experimental configuration, specifically, the channel height, so the drag reduction is not an ideal quantity to characterize the drag reduction performance of a given surface and to be compared with published results. With the experimental measurements, the effective slip length can be computed using the method described in Section 3.2.4, and the results are presented in Figure 4.8. The effective slip length has a similar dependence on the mass flow rate as drag reduction, but the spread of the effective slip length is enlarged. That is because the calculation method uses drag reduction ratio as a parameter and is sensitive to the variance of the input parameters, such as pressure loss and channel height.

Figure 4.8: The effective slip length $b_{\text{eff}}$ as a function of the mass flow rate $\dot{m}$ when the lubrication cavity is filled with air.
In Figure 4.8, the effective slip lengths extracted from experimental data by the 2D model are plotted with a dashed line, and the effective slip lengths extracted from experimental data by the 3D model are plotted with a solid line. As mentioned in Chapter 3, the 2D model does not account for the no-slip side walls effects but the 3D model does, making the effective slip length extracted with the 3D model larger than the effective slip length extracted with the 2D models.

Only a few researchers observed such a large effective slip length in experiments. Srinivasan et al. [58] measured the effective slip length of the dual-textured spray-coated meshes with a rheometer and reported an effective slip length range from 94 $\mu$m to 213 $\mu$m. Lee et al. [36] achieved an effective slip length larger than 20 $\mu$m and up to 185 $\mu$m by fabricating microstructures of posts and grates on a Teflon coated silicon wafers, the effective slip length of which were measured with a rheometer.

### 4.6 Simulation conditions

In this work, two models were developed: a full-scale model of the experimental system to have a better understanding of the experimental results and a unit cell model to reveal the pattern-dependent influence on the flow behavior adjacent to the surface.

To model the air-filled cases, the approach of Samaha [26] was followed and zero shear force was prescribed on the air-water interfaces. The air-water interface was assumed to be flat in both the full-scale and the unit cell models for simplicity, excluding models that were developed with the approach of Karatay [4] to investigate the effect of the air-water interface shape on the effective slip length.

All the simulations in the present work focused on the steady, incompressible, laminar flow of water. The Navier-Stokes equations were numerically solved using COMSOL Multiphysics, which is a simulation software based on the finite element method. The mesh was physics-controlled (non-uniform and adaptive) and automatically generated by COMSOL Multiphysics. To ensure the results were independent of the choice of mesh size, at least two grid sizes were used in the simulations.
4.6.1 The full-scale model

For the full-scale model, the dimensions of the simulation domain were determined by the dimensions of the flow cell and perforated sheets in the experimental setup, such as the channel height, the channel width, the hole size, and the spacing between holes. However, to save computing time, the length of the channel was shortened. To prevent losing the physical details in the region of interest, the flow must be fully developed before it arrives in the test region. As a result, the distance from the inlet to the hole array should be larger than the entrance length. The entrance length is given by $L_{h,\text{ laminar}} = 0.0575ReD_h$ [59] for laminar flow, can provide a good reference for shortening the length of the channel. For the highest Reynolds number, the estimated entrance length was $L_{h,\text{ laminar}} = 36.9 \text{ mm}$.

Figure 4.9: Schematic diagram of the full-scale model.

For the full-scale model, the inlet flow is uniform at $u = (\dot{m}/\rho)/(WH)$. At the outlet, constant pressure of $p = 0 \text{ Pa}$ was prescribed. The no-slip boundary condition was applied on the inner surface of the simulation domain excluding the area representing the air-water interface. Figure 4.9 illustrates the full-scale model that was used to investigate the bypassing flow behavior. Note that the length between the perforated region to the outlet is shortened compared with that of the flow cell because the pressure distribution and velocity field further downstream are not of interest.

The full-scale model was used to give: (1) pressure loss over the reference surface and test surfaces; (2) pressure distribution along the midline parallel to the flow direction on the upper no-slip wall; (3) the effects of the depth of the cavity on the pressure distribution for
the water-filled cavity.

4.6.2 The unit cell model

For the unit cell model, the dimensions of the simulation domain were determined by the dimension of the unit cell of the surface design. If the unit cell is a triangle, the rectangular simulation domain only needs to cover half of the unit cell divided by the symmetry axis parallel to the flow direction.

For the unit cell models, the inlet and outlet conditions are periodic conditions with a prescribed pressure drop $\Delta p$. The symmetry boundary condition is applied to the sidewalls of the simulation domain. The no-slip boundary condition is applied to the rest of the inner surface of the computational domain excluding the area that represents the air-water interface. Figure 4.10 illustrates the configuration of a unit cell model.

![Figure 4.10: Schematic diagram of the unit cell model.](image-url)
The unit cell model was used to investigate: (1) the effects of the air-water interface shape on the effective slip length, which will be presented in Section 4.7.2; (2) the pattern-dependent effects of the flat air-water interface on the effective slip length, which will be presented in Section 4.7.3.

4.7 Simulation results

4.7.1 The effects of bypass flow on drag reduction

As mentioned in Section 4.5.2, the drag reduction observed in water-filled cases was not only produced by the slip surface (liquid-liquid interface) but also by water bypass through the lubrication cavity. To have more insight into the bypass flow behavior, a full-scale model was developed with the setting described in Section 4.6.1.

Figure 4.11: The pressure distribution along the flow direction. In this case, the cavity depth is 4 mm, and the uniform inlet velocity is 2.8842 m/s.
Figure 4.11 shows the pressure distribution at the middle line of the upper wall of the channel along the flow direction. The pressure gradient over the perforated sheet (|\(k_{\text{test}}\)|) was much smaller than that over the flat sheet (|\(k_{\text{ref}}\)|). A significant pressure loss at the end of the perforated sheet was observed, which was due to the bypass flow exiting the lubrication cavity. Also, the velocity field and pressure field of this full-scale simulation indicated that the flow was fully developed within a distance of 20 mm measured from the inlet. In the following models, the length of the full-scale model was shortened to reduce the computational time.

Figure 4.12: The pressure distribution along the flow direction at the different uniform inlet velocity. In this case, the cavity depth is 4 mm.
The pressure distribution for different uniform inlet velocities (corresponding to different mass flow rates) are summarized in Figure 4.12. The pressure loss due to the bypassing flow exiting the cavity increases as the mass flow increases, which explains why the drag reduction decreases as the mass flow rate increases when the cavity is filled with water. Also, the pressure gradient of the linear region over the perforated sheet does not change significantly within the test range of the mass flow rate.

![Figure 4.13: The pressure distribution along the flow direction computed with different cavity depth. In this case, the uniform inlet velocity is 2.8842 m/s.](image)

The pressure distribution computed with different cavity depths is summarized in Figure 4.13, which shows that the pressure gradient increases as the cavity depth decreases due to increased flow resistance. As the cavity depth decreases, the cavity length-to-depth increases.
so that the cavity becomes a closed cavity [60]. Therefore, the shear layer of the bypass flow attaches to the cavity floor when the cavity depth is 1 mm, which explains why the pressure distribution curve of the one-millimeter cavity depth has a significantly larger slope and crosses over other curves in Figure 4.13.

4.7.2 The effects of the air-water interface shape on the effective slip length

Following the approach described in Section 4.6.2, with \( d = 1 \) mm, \( s = 1.1 \) mm, and \( H = 2 \) mm, a unit-cell model was developed. The air-water interface was assumed to be part of a sphere and characterized by protrusion angle \( \theta \). Inspired by Karatay’s [4] idea, a prescribed velocity boundary condition is applied to the upper wall of the simulation domain. The simulation domain and shape of the air-water interfaces are shown in Figure 4.14.

Figure 4.14: The shape of the air-water interface in the unit cell simulation domain: (a) Convex; (b) Concave. The color bar refers to the velocity magnitude, which is given in meters per second.
With the unit cell model, the pressure difference between the inlet (upstream) and outlet (downstream) can be calculated from the average pressure at the inlet and outlet. Using the model developed in Chapter 2, the effective slip length can be calculated. Note that the sidewalls of the simulation domain are assumed to be symmetric, so the 2D model should be used to calculate the effective slip length.

Figure 4.15: The effective slip length $b_{\text{eff}}$ as a function of protrusion angle $\theta$.
In Figure 4.15, the effective slip length is plotted as a function of protrusion angle, which shows that the effective slip length can vary significantly as the shape of the air-water interface changes. Within the range of $-60^\circ < \theta < 60^\circ$, as $\theta$ increases, the effective slip length first increases to its maximum at $\theta \approx 8^\circ$, then decreases and drops below zero at $\theta \approx 48^\circ$. The negative effective slip length indicates that the corresponding no-flat air-water interface increased the drag. To achieve the high-precision measurement of the effective slip length, it is necessary to have a pressure control system to keep the shape of the air-water interface unchanged during the measurement process.

4.7.3 The effects of the surface design on the effective slip length

In Figure 4.16, the schematic diagram of twelve surface designs is presented, and the unit cell of each surface design is marked in a red dashed line. In addition, the air-water interface is assumed to be flat i.e. $\theta = 0^\circ$.

These twelve surface designs were investigated with the proposed method. The height of the simulation domain of the unit cell model was 100 $\mu$m. The width and length of the domain were determined by the porosity and feature size. Specifically, the porosity was 75%, the width of the no-shear grooves was 75 $\mu$m for case 1 and case 2, the diameter of the holes or posts was 15 $\mu$m for case 3 to case 8, and the diameter of the inscribed circle of the hexagonal holes or posts was 15 $\mu$m for case 9 to case 12. The corresponding effective slip lengths are presented in Figure 4.17. Four of the designs have analytical formulas available, which can be used to validate the calculated results. In addition, Figure 4.17 clearly shows that the effective slip length is a surface characteristic and independent on the flow configuration.

Effective slip length is one of the metrics determining drag reduction, which also characterizes surface friction. Since a surface with smaller average surface friction will have a larger effective slip length, the goal is to optimize the surface design to achieve a larger effective slip length.
Figure 4.16: Schematic diagram of the studied patterns, which are labeled from case 1 to case 12. In these cases, the no-slip area is marked in gray while the no-shear area is marked in white. Their geometry and layout are described by a unit cell (marked with the red dashed line) manner. (1) no-shear groove parallel to the flow direction; (2) no-shear groove transverse to the flow direction; (3) no-slip circle on a rectangular lattice; (4) no-shear circle on a rectangular lattice; (5) no-slip circle on a hexagonal lattice with vertical rows; (6) no-slip circle on a hexagonal lattice with horizontal rows; (7) no-slip hexagon on a hexagonal lattice with vertical rows; (8) no-slip hexagon on a hexagonal lattice with horizontal rows; (9) no-shear circle on a hexagonal lattice with vertical rows; (10) no-shear circle on a hexagonal lattice with horizontal rows; (11) no-shear hexagon on a hexagonal lattice with vertical rows; (12) no-shear hexagon on a hexagonal lattice with horizontal rows.
Figure 4.17: Slip length estimation for different patterns with a fixed porosity (75%) using the published analytical formula listed in Table 1 (red circle) and our CFD-based method with pressure-driven (blue filled circle) or wall-driven flow (black circle) assumption. The studied patterns are labeled from case 1 to case 12, and their geometry and layout have been illustrated and described in Figure 4.16.

With the method developed, various patterns can be theoretically investigated, and their effective slip length can be estimated. In this way, the theoretical optimal design (i.e. the design with the largest estimated effective slip length) can be found. Although some physics were ignored (e.g. gas diffusion, surface tension, etc.) in the method proposed, it can be used to roughly assess the surface design in a screening process. If a surface design has a small estimated effective slip length, it will be screened out. The patterns with large estimated effective slip lengths can be fabricated on polyimide sheets with the laser machining system described in Section 3.1.2 so that their effective slip lengths can be experimentally measured with the configuration described in Section 3.2.6. and the optimal surface design identified.
Chapter 5
CONCLUSIONS AND FUTURE WORK

5.1 Summary

This dissertation focused on designing and fabricating perforated sheets used in conjunction with underlying cavities for providing effective slip flow and characterizing their drag reduction effects. A detailed investigation has been conducted, including theoretical analysis, numerical modeling, and experimental testing.

As discussed in Chapter 2, an analytical solution for duct flow with general boundary conditions was derived, which allowed the sidewall effects to be included in duct flow analysis. Also, an analytical solution for flow between two parallel plates with general boundary conditions was derived. These two analytical solutions can be used to solve for the effective slip length when the pressure gradient and average velocity are known. As detailed in Section 3.2.4, an alternative way to measure the effective slip length when the pressure loss or the flow rate is fixed was proposed.

As shown in Section 3.1.1, an improved design was proposed for the surface gas/liquid entrapment based on the previously published results. Then, the control software for the laser micromachining system was described in Section 3.1.2, so that the design could be successfully fabricated on a polyimide sheet. To measure the drag reduction effects of the perforated sheets, a flow cell system was designed, built, and refined as described in Section 3.2.6.

5.2 Conclusions

The analytical model developed in Chapter 2 revealed that, when the aspect ratio is less than 10, the normalized velocity profile and normalized average velocity predicted by the 3D model will significantly differ from that predicted by the 2D model due to the side-wall effects. However, the excellent agreement between the 3D model analysis and 2D model
analysis for a very large aspect ratio \((w/h \approx 100)\) indicates that the 2D model is accurate in this regime.

Different perforated sheets were evaluated in channel flow with an air-filled cavity or a water-filled cavity at different mass flow rates. Within the mass flow rate range of \(1.382 \text{ g/s} \sim 2.764 \text{ g/s}\), large drag reduction was measured experimentally. Specifically, the drag reduction observed in water-filled cases was up to 66% and much higher than expected. Based on the video of the dye experiments, bypass flow was observed. The simulation results provided quantitative evidence to prove that such a large drag reduction was due to the bypass flow through the lubrication cavity. As for the air-filled cases, the air can be effectively trapped in the holes, and the air-water interface can resist the water pressure and viscous shear forces even at the highest mass flow rate, 2.764 g/s.

The drag reduction reported in this work is from 22% to 34% for air-filled cases, which corresponds to the effective slip length from 90 \(\mu m\) \(\sim\) 180 \(\mu m\). These results are an improvement over the drag reduction observed by Hao et al. \[33\], 10% \(\sim\) 30%, and similar to that reported in Ou’s paper \[34\], up to 40%. However, in these studies, drag reduction was achieved by introducing micro-scale and nano-scale structures, which required clean-room techniques to fabricate, and limited the application of their approach. In this work, the large drag reductions were obtained on such surfaces that have a feature size of 600 \(\mu m\) \(\sim\) 1000 \(\mu m\) so that conventional tools can be utilized for fabrication.

As demonstrated in Section 4.1, the channel height can significantly change the drag reduction when the surface has the same slip effect, as characterized by \(b_{\text{eff}}\). Therefore, the comparison of the effective slip length can better reflect the drag reduction effect. Compared to the effective slip length with \(b_{\text{eff}} \approx 2 \mu m\) by Hao et. al. \[33\] and \(b_{\text{eff}} \approx 20 \mu m\) by Ou et. al. \[33\], the effective slip length presented in this work is much better.

Only a few researchers observed an effective slip length as large as this work, such as 94 \(\mu m\) \(< b_{\text{eff}} < 213 \mu m\) obtained over a dual-textured spray-coated meshes by Srinivasan et. al. \[58\], and 20 \(\mu m\) \(< b_{\text{eff}} < 185 \mu m\) achieved on the Teflon coated surface by Lee et. al. \[36\]. However, achieving a large effective slip length with the approach described in this work does not require chemical coating, which is favored by many application scenarios.
Computational fluid dynamics (CFD) models were developed to compute the pressure distribution for water-filled cases to provide more insight into the bypass flow behavior. Also, the CFD models were used to calculate the pressure gradient and average velocity for a flow over a patterned surface, so that the effective slip length of the patterned surface can be found with the analytical solutions developed in Chapter 2. With this approach, the effect of the air-water interface shape on the effective slip length was found, that is, the effective slip length increases to its maximum at $\theta \approx 8^\circ$, then decreases and drops below zero at $\theta \approx 48^\circ$ as $\theta$ increases from $-60^\circ$ to $60^\circ$. As a result, it is not only required to introduce more slip-surface area but also to maintain a proper protrusion angle, to achieve a larger effective slip length. In addition, the effects of the surface design on the effective slip length are discussed in Section 4.7.3. With the same porosity and feature size, the effective slip length can be different due to the differences in the shape and the layout of the patterns.

5.3 Future work

There are still several topics that need further investigation.

5.3.1 More robust surface gas/liquid entrapment

Corresponding to the surface gas or liquid entrapment, the lubrication cavity can be filled with either liquid (e.g. water) or gas (e.g. air), respectively. When the lubricant in the cavity differs from the fluid flowing in the channel, the interfaces between the lubricant and the fluid should be properly maintained to avoid lubricant leakage into the channel, which requires the tension (surface tension for the gas lubrication or the interfacial tension for the liquid lubrication), the fluid pressure (i.e. pressure in the channel) and the lubrication pressure (i.e. pressure in the lubrication cavity) to balance. Note that the fluid pressure is usually determined by the application scenario, whereas the lubrication pressure can be actively controlled, and the tension can be manipulated by chemically modifying the surface energy (e.g. using superhydrophobic coating) or changing the surface design because the tension depends on the shape of the interface. Therefore, a customized surface design and actively controlled lubrication cavity can be utilized to maintain fluid-lubricant interfaces so that a more robust surface gas or liquid entrapment can be achieved.

The pressure gradient associated with the fluid motion increases the fluid pressure from
upstream to downstream, whereas the lubrication pressure is almost constant everywhere because the lubricant trapped in between the surface structures is coupled to the same lubrication cavity. When the tension on the fluid-lubricant interface cannot compensate for the fluid pressure change, the lubricant could leak into the channel. To avoid this, multiple lubrication cavities should be used and pressurized differently from upstream to downstream.

To achieve the high precision measurement of the effective slip length, it is necessary to keep the air-water interface stable and unchanged during the measurement process so that a real-time active control system is needed for the lubrication cavity. The system is expected to respond to any ambient changes, such as pressure, temperature, and shear forces that might affect the shape of the air-water interface.

5.3.2 Alternative methods to measure drag reduction effect

The experimental methods used in this dissertation rely on measuring the pressure loss over the surface. However, compared with the environmental noises, the pressure loss is small and the difference between pressure loss over the test surface and the pressure loss over the reference surface is also small. This makes the experimental investigation challenging. To resolve these challenges, a larger perforated sheet or alternative experimental methods, such as µPIV, could be used.

5.3.3 External boundary layer drag reduction

In this dissertation, all the theoretical analysis and experimental studies focused on the fluid flow in a rectangular duct i.e. internal flow, so that the drag reduction performance of the perforated sheet in external flow is still unknown. Martin [61] introduced a slip length condition to the boundary layer problem, and obtained a Blasius boundary layer solution with slip flow conditions, which can provide a physical picture of the slip effect in the boundary layer flow. Using the Blasius boundary layer solution with slip flow conditions, Martin and Boyd [62] investigated the momentum and heat transfer in a laminar boundary layer with slip flow. Then, Martin and Boyd [63] published the Falkner–Skan solution with slip flow conditions. The slip effect can also be represented by a constant slip velocity at the boundary. Due to the simplicity of the constant slip velocity condition, the Von Kármán integral method can provide an estimate of the friction at the boundary.
If the experimental conditions permit, testing the perforated sheets in an external flow configuration would be the ideal next step. Elbing's [6] works can be a good reference for designing the experiments and analyzing experimental data.
APPENDIX A: THE MANUAL FOR CREATING A FABRICATION INPUT FILE

The laser micromachining system requires an input file to read in order to know the pattern to cut. The input files have the same filename extension .esp300.

A .esp300 file requires:

1. A begin statement in the first line of the .esp file, formatted as: BEGIN

2. An end statement in the last line of the .esp file, formatted as: END

3. To make a comment, start with ! on a new line. Any comments will be displayed in the command line while the software is running.

4. The .esp300 file requires that certain parameters are defined. These parameters are:

   (a) Z: z-location of the motion stage (determined based on the focus of the laser for a given thickness of a sheet to be cut)

   (b) F: the laser repetition frequency in Hz,

   (c) E: the feed current for the laser as a percentage

   (d) V: the velocity of the motion stage in mm/s

   (e) A: the acceleration of the motion stage in mm/s.

   (f) P xp,yp: moves to a point (xp,yp) with the laser off, where xp, yp are in the unit of mm

   (g) L xp,yp: moves to a point (xp,yp) with the laser on, where xp, yp are in the unit of mm

   (h) C xc,yc,d: moves in an arc centered at xc,yc for d degrees with the laser on, where xc, yc are in the unit of mm

   (i) B n: initiates burst mode if 0 < n ≤ 4000; if n = 0, burst mode is disabled
(j) D: emits a burst of n pulses, where n is defined when burst mode is initiated

(k) R: reset the laser operation mode and status

(l) LASER ON: turn on the laser

(m) LASER OFF: turn off the laser

(n) SL xp,yp: moves to a point (xp,yp) without changing the laser operation status,
where xp, yp are in the unit of mm

(o) SC xc,yc,d: moves in an arc centered at (xc,yc) for d degrees without changing
the laser operation status, where xc, yc are in the unit of mm

EXAMPLE:

! Begin statement
BEGIN
! Set Z location
Z 13.60000
! Set repetition frequency for laser (Hz)
F 1000
! Set feed current for laser (%)
E 50.00
! Set motion speed (mm/s)
V 0.05000
! Set acceleration (mm^2/s)
A 20.00000
! Move to (0.0)
P 0.000000,0.000000
! Cut a line from (0.0) to (2.5,0)
L 2.500000,0.000000
! Cut a 2.5mm diameter circle centered at (5.0)
C 5.000000,0.000000,360
! Move to (5,5) without changing the laser current status (i.e off)
SL 5.000000,5.000000
! Turn on the laser
LASER ON
! Move to (0,5) without changing the laser current status (i.e. on)
! Cut a line from (5,5) to (0,5)
SL 0.000000,5.000000
! Move to (-1,0) without changing the laser current status (i.e. on)
! Cut a line from (0,5) to (-1,0)
SL -1.000000,0.000000
! Move moves in an arc centered at (0,0) for d degrees
! without changing the laser current status (i.e. on)
! Cut a 1,0mm diameter circle centered at (0,0)
SC 0.000000,0.000000,360
! Turn off the laser
LASER OFF
! Set burst mode
B 1
! Move to (2.5,2.5)
P 2.500000, 2.500000
! Trigger one burst emission
D
! Reset laser status to the default setting
R
! End statement
END
APPENDIX B: MATLAB SOURCE CODE

Script for calculating Laplace pressure

```matlab
fclose all
close all
clear all
clc

%% Laplace pressure
gamma = 72.86e-3;% surface tension, unit: N/m

%%
figure
hold on
d = 1.0e-3; % diameter of the hole, unit: m
a = d/2; % radius of the hole, unit: m
R = a./sind(-90:5:90); % radius of the curvature, unit: m
% pressure difference across the interface
% i.e. interior pressure minus the exterior pressure
Delta_p = 2*gamma./R;
plot(-90:5:90,Delta_p);

% diameter of the hole, unit: m
% radius of the hole, unit: m
% radius of the curvature, unit: m
% pressure difference across the interface
% i.e. interior pressure minus the exterior pressure
Delta_p = 2*gamma./R;
plot(-90:5:90,Delta_p)
```

83
d = 0.6e-3; % diameter of the hole, unit: m
a = d/2; % radius of the hole, unit: m
R = a./sind(-90:5:90); % radius of the curvature, unit: m
% pressure difference across the interface
% i.e. interior pressure minus the exterior pressure
Delta_p = 2*gamma./R;
plot(-90:5:90,Delta_p)

xlabel('\theta(\circ)')
ylabel('\DeltaP_{Laplace} (Pa)')
box on
grid on
legend({'d = 1.0 mm', 'd = 0.8 mm', 'd = 0.6 mm'}, 'Location', 'northwest')
BIBLIOGRAPHY


[19] ——, “Wetting and Active Dewetting Processes of Hierarchically Constructed
Superhydrophobic Surfaces Fully Immersed in Water,” Journal of

“Continuous flow switching by pneumatic actuation of the air lubrication layer on
superhydrophobic microchannel walls,” in 2008 IEEE 21st International Conference

“Pressure-Stable Air-Retaining Nanostructured Surfaces Inspired by Natural Air
Plastrons,” Advanced Materials Interfaces, vol. 5, no. 13, p. 1800125, 2018, publisher:
Wiley Online Library. 3

Available: http://www.nature.com/articles/nmat1962 3, 41

Lubricant-Impregnated Surfaces in Viscous Laminar Flow,” Langmuir, vol. 30, no. 36,
https://pubs.acs.org/doi/10.1021/la5021143 3

Available: http://link.springer.com/10.1007/s00348-016-2264-z x, 3, 4, 5, 8, 9, 10

of pdms/hydrophobic silica superhydrophobic coating for drag reduction application,”
https://linkinghub.elsevier.com/retrieve/pii/S0257897220310975 3, 4, 10

[26] M. A. Samaha, H. Vahedi Tafreshi, and M. Gad-el Hak, “Modeling drag reduction and
meniscus stability of superhydrophobic surfaces comprised of random roughness,”

[27] D. Byun, J. Kim, H. S. Ko, and H. C. Park, “Direct measurement of slip flows in
superhydrophobic microchannels with transverse grooves,” Physics of Fluids, vol. 20,
no. 11, p. 113601, Nov. 2008. [Online]. Available:

drag-reducing ultrahydrophobic surfaces,” Physics of Fluids, vol. 17, no. 10, p. 103606,
2005. [Online]. Available:
http://scitation.aip.org/content/aip/journal/pof2/17/10/10.1063/1.2109867 3, 5, 10

87


89


