Mathematical Modelling of
Particle-fluid Flows
in Microchannels

Kittisak Chayantrakom

This thesis is presented for the Degree of
Doctor of Philosophy of
Curtin University of Technology

January 2009
Declaration

To the best of my knowledge and belief this thesis contains no material previously published by any other person except where due acknowledgment has been made.

This thesis contains no material which has been accepted for the award of any other degree or diploma in any university.

Kittisak Chayantrakom

21 January, 2009
Acknowledgments

I would like to express my gratitude to my supervisor, Professor Yong Hong Wu, for his valuable guidance. His helpful suggestions and encouragement have been of great value to me in the research and the preparation of this thesis. I would also be grateful to my associate supervisor, Associate Professor Benchawan Watanapataphee, for her helpful advices and suggestions.

I would like to thank Royal Thai Government for the financial support during the period of my study. I would also like to thank Mahidol University for providing me the opportunity to study in Perth. I deeply appreciate the Office of Education Affairs, Royal Thai Embassy in Canberra for all their kind assistance during my study.

Finally, I am so thankful to the Department of Mathematics and Statistics, Curtin University of Technology for providing me with all the necessary facilities for my research. I also appreciate all the staff and students in the Department of Mathematics and Statistics for their friendship and encouragement during my study at the University.
Abstract

Flows of fluids and solid particles through microchannels have a very wide range of applications in biological and medical science and engineering. Understanding the mechanism of microflows will help to improve the development of the devices and systems in those applications. The aim of this study is to develop a sophisticated simulation and analysis technique for the study of fluid-particle flow through microchannels. This work involves construction of mathematical models, development of analytical methods and numerical algorithms, and numerical investigation and analysis.

The study consists of three parts. The first part of the research focuses on the transient flow of an incompressible Newtonian fluid through a micro-annual with a slip boundary. The flow of the fluid is governed by the continuity equation and the Navier-Stokes equations, and is driven by the pressure field with a time-varying pressure gradient. By using the Fourier series expansion in time and Bessel functions in space, an exact solution is derived for the velocity field. The velocity solution is then used to obtain the exact solutions for the flow rate and the stress field. Based on the exact solutions, the influence of the slip parameter on the flow behaviour is then investigated.

The second part of the research focuses on the particle-fluid flow in microchannels. The transport of fluid in the vessel is governed by the continuity equation and the transient Navier-Stokes equations, while the motion of the particles is governed by Newton’s laws. The particle-wall and particle-particle interactions are modelled by the interacting forces, while the particle-fluid interaction is described
by the fluid drag force. A numerical scheme based on the finite element method
and the Arbitrary Lagrangian-Eulerian method is developed to simulate the motion
of the particles and the fluid flow in the vessels. The influence of boundary slip on
the velocity field in the fluid is also investigated numerically.

Based on the work in the second part, the third part of the research focuses on
the control of the movement of particles in the fluid by applying an external mag-
netic field to the system. Maxwell’s equations are used to model the magnetic field
generated by the external magnetic source, and a finite element based numerical
scheme is developed to solve the underlying boundary value problem for the mag-
netic flux density generated. From the computed flux density and magnetic vector
potential, the magnetic forces acting on the particles are determined. These mag-
netic forces together with the drag force and the particle-particle interacting forces
dominate the behaviour of the particle motion. A numerical scheme, similar to that
for the second part of the research, is then developed to study the fluid-particle flow
in microchannels under magnetic forces, followed by a numerical investigation on
the influence of the magnetic forces on the particle flow behaviour.
List of Publications Related to This Thesis


Contents

Declaration i

Acknowledgments ii

Abstract iii

List of Publications Related to This Thesis v

List of Figures xi

List of Tables xii

Nomenclature xiii

1 Introduction 1

1.1 Background ........................................... 1
1.2 Objectives ........................................... 3
1.3 Outline of the Thesis ................................. 4

2 Literature Review 6

2.1 General Overview ................................... 6
2.2 Modelling of Magnetic Field ......................... 6
2.3 Modelling of Fluid Flow ............................. 12
2.4 Modelling of Particle-fluid Flow ..................... 17
2.5 Concluding Remarks ................................ 20
List of Figures

3.1 The coordinate system used .......................................................... 23
3.2 Variation of the flow rate with \( l \) and \( k \) obtained from solution (3.35)  
with \( \lambda = 1 \): (a) 3D graph for \( Q^*(k, l) \); (b) contour plot of \( Q^*(k, l) \)  
on \((k, l)\) plane. ................................................................. 31
3.3 Variation of the amplitude of the flow rate \( Q_m^* \) with \( l \) and \( k \) obtained  
for different frequencies \( \omega = \alpha \mu/\rho R^2 \) with (a) \( \alpha = 0.005 \), (b)  
\( \alpha = 0.00025 \), and (c) \( \alpha = 0.00005 \). ..................................... 33
3.4 Influence of \( l \) on the quasi steady-state solution \( Q_s^* \) and the transient  
solution \( Q_m^* \) under different frequencies \( \omega = \alpha \mu/\rho R^2 \) with five  
different \( \alpha \) values: \( \alpha = 5.0 \times 10^{-3} \) (dash-box line), \( \alpha = 1.0 \times 10^{-3} \)  
(dash-circle line), \( \alpha = 5.0 \times 10^{-4} \) (dash-diamond line), \( \alpha = 2.5 \times  
10^{-4} \) (dash-cross line), \( \alpha = 5.0 \times 10^{-5} \) (dash line). ................. 34
3.5 Variation of the amplitude of the flow rate with \( l \) and \( k \) obtained  
from solution (44) with \( \lambda = 1 \), \( R = 1.0 \times 10^{-5} \), \( \rho = 1060 \), \( \mu =  
0.001 \), \( \omega = 5.0 \times 10^{-4} \mu/\rho R^2 \): (a) 3D graph for \( Q_m^* \); (b) contour  
plot of \( Q_m^* \) on \((k, l)\) plane. .................................................... 35
3.6 Variation of the amplitude of the flow rate with \( l \) for \( \omega = 5.0 \times  
10^{-4} \mu/\rho R^2 \) and various different \( k \) values in case 2: \( k = 0.1 \) (solid  
line), \( k = 0.3 \) (dash line), \( k = 0.5 \) (dash-cross line), \( k = 0.7 \) (dash-box  
line), and \( k = 0.9 \) (solid-circle line). .................................... 36
3.7 Variations of the amplitude of the slip velocity $u_m^*$ on the outer surface $r^* = 1$ with $l$ for $\omega = 5.0 \times 10^{-4} \, \mu/\rho R^2$ and various different $k$ values in case 2: $k = 0.1$ (solid line), $k = 0.3$ (dash line), $k = 0.5$ (dash-cross line), $k = 0.7$ (dash-box line), and $k = 0.9$ (solid-circle line).

4.1 The computation domain and its finite element mesh together with the coordinates.

4.2 Cross-section profile of the velocity $u_x$ at three different locations along the channel for five different slip parameter values $l = 0$ (or no slip) (solid-box line), $l = 0.25 \times 10^{-7}$ (solid-cross line), $l = 0.5 \times 10^{-7}$ (solid-asterisk line), $l = 1.0 \times 10^{-7}$ (solid-circle line), and $l = 2.0 \times 10^{-7}$ (solid-triangle line): (a) at the location $x = 0.5 \mu m$, (b) at the location $x = 1.5 \mu m$ (through the center of the microcylinder), and (c) at the location $x = 2.5 \mu m$ along the microchannel. The $x$ and $y$ coordinates are shown in Figure 4.1.

4.3 Influence of the slip parameter $l$ on the drag force exerted on the cylinder by fluid. The circles represent numerical results, while the solid line is the least square fitting of the numerical results.

4.4 Influence of the slip parameter $l$ on the flow rate of fluid through the microchannel.

4.5 Streamlines and vector plots of the velocity fields in fluid in the microchannel for five different slip parameter values: (a) $l = 0$ (no slip), (b) $l = 0.25 \times 10^{-7}$, (c) $l = 0.5 \times 10^{-7}$, (d) $l = 1.0 \times 10^{-7}$, and (e) $l = 2.0 \times 10^{-7}$. The scaled bar shows the scale of the magnitude of velocity in m/s.

4.6 Velocity profiles at various instants of time for the case with one particle.
4.7 Velocity profiles at various instants of time for the case with three particles. .................................................. 57
4.8 Velocity profiles at various instants of time for the case with five particles. .................................................. 57
4.9 Velocity profiles at various instants of time for the case with nine particles. .................................................. 57
4.10 Pressure profiles along the flow direction at \( t = 0 \)s. ............... 58
4.11 Computed results for the fluid-particle flow with \( V_p(t_0) = 0 \): (a)–(b) the \( x \) and \( y \) coordinates of the locations of the particle; (c)–(d) the \( x \) and \( y \) components of the particle velocity along the channel; (e)–(f) the \( x \) and \( y \) components of the total force acting on the particle; (g) the trajectory the of particle. ................................. 59
4.12 Computed results for the fluid-particle flow with \( V_p(t_0) = u(t_0) \): (a)–(b) the \( x \) and \( y \) coordinates of the locations of the particle; (c)–(d) the \( x \) and \( y \) components of the particle velocity along the channel; (e)–(f) the \( x \) and \( y \) components of the total force acting on the particle; (g) the trajectory the of particle. ................................. 60
4.13 Computed results for the fluid-particle flow with \( V_p(t_0) = 2u(t_0) \): (a)–(b) the \( x \) and \( y \) coordinates of the locations of the particle; (c)–(d) the \( x \) and \( y \) components of the particle velocity along the channel; (e)–(f) the \( x \) and \( y \) components of the total force acting on the particle; (g) the trajectory the of particle. ................................. 61
5.1 The computation domain, the location of magnetic source, and the finite element mesh. ................................. 68
5.2 Velocity profiles and particle locations at various instants of time for the case with three magnetic particles, shown together with the contour lines of the magnetic potential \( A_z \). ................................. 69
5.3 Magnetic forces acting on the particles for the three particles case: (a) the $x$-component of the magnetic force; and (b) the $y$-component of the magnetic force. The solid-triangle line is for particle 1; the solid-box line is for particle 2; and the solid-plus line is for particle 3.

5.4 The location of the particles as a function of time for the three particles case: (a) the $x$-coordinate; (b) the $y$-coordinate. The solid-triangle line is for particle 1; the solid-box line is for particle 2; and the solid-plus line is for particle 3.
List of Tables

2.1 Blood viscosity models, $\mu$, given in Poise (P) as a function of strain rate, $\dot{\gamma}$, given in $s^{-1}$ [42]. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . 15
Nomenclature

A  magnetic vector potential
B  magnetic flux density
F_q collision force of the qth particle
F_{q,p} particle-particle interaction force
F_{q,w} particle-wall interaction force
F_{mag} magnetic force acting on the particles
F_s fluid drag force acting on the particles
H  magnetic field strength
J  Jacobian matrix
J_0, J_1 zero-order and first order Bessel functions of the first kind
L_{center} entrance length of the fictitious inlet for laminar flow
M  magnetization of the magnetic particles
Q  flow rate of fluid flux
R  outer radius of annulus
R_q radius of the qth particle
V_q velocity vector of the qth particle
X  coordinates of the reference system
X_q center of the qth particle
Y_0, Y_1 zero-order and first order Bessel function of the second kind
d  rate of deformation tensor
d_{q,p} distance between the centers of the qth and the pth particle
d_{q,w} distance between the centers of the qth particle and the imaginary particle on the other side of the wall
l  normalized slip parameter
l_1 slip parameter of the surface of the inner annulus
l_2 slip parameter of the surface of the outer annulus
### Mathematical Symbols

- \( m_q \): mass of the \( q \)th particle
- \( p \): pressure of fluid
- \( P_{\text{enter}} \): entrance pressure at the channel inlet
- \( t \): time
- \( u \): velocity vector of fluid in annulus
- \( u_i \): velocity component of fluid in the \( i \)th-direction
- \( u_{\text{mean}} \): average velocity of the laminar flow
- \( v_{\text{st}} \): velocity component of the solid surface on the tangential direction
- \( v_t \): velocity component of the fluid on the tangential direction
- \( x \): coordinates of the moving coordinate system
- \( \varepsilon_q \): stiffness parameter of the \( q \)th particle
- \( \varepsilon_w \): stiffness parameter of the wall
- \( \Omega \): spatially fixed domain
- \( \Omega_{\text{de}} \): deformed domain
- \( \Omega_q \): domain of the \( q \)th particle
- \( \Psi \): mesh velocity
- \( \mu, \mu_f \): viscosity of fluid
- \( \mu_r \): relative permeability of a magnetic material
- \( \mu_0 \): permeability in vacuum
- \( \rho, \rho_f \): density of fluid
- \( \sigma_{ij} \): stress-deformation rate
Chapter 1

Introduction

1.1 Background

Recent advance in microtechnologies has led to the development of many biological and engineering devices and systems in microscales. Typical examples include biochemical lab-on-the-chip systems, micro-electromechanical systems (MEMS), fuel cell devices, biological sensing and energy conversion devices [58], and drug delivery systems [76]. Most of these devices and systems involve flows of fluids and particles in microchannels, referred to as microflows [10, 33, 35, 37]. Since the functional characteristics and physical phenomena of the systems are determined by the behaviour of fluid and particle flow in these systems, it is extremely important to study the mechanism of microflows in order to gain better understanding [5, 28, 85].

In recent years, many research projects involving fluid or particle flow study have been undertaken. In the lab-on-the-chip application, it has been reported that the DEPArray(tm) Chip can be used to capture and control movement of lipid microparticles [80]. Microfluidic devices integrated with a DNA computer model can be used for medical diagnostics and drug screening [49]. Nanocrystalline diamond with Chromatin immunoprecipitation protocols (ChIP) can be used for analyzing DNA [43]. Magnetic beads are separated from fluid in microfluidic channels using microelectromagnets [72]. For the MEMS application, the microthermal preconcentrators ($\mu$TPCs) is used for gas analysis of volatile organic compounds (VOCs) [2]. A MEMS viscometer is used in the oil well logging process by an-
alyzing the density and viscosity of fluids [26]. A microfluidic device consisting of MEMS-fabricated channels is used to produce microbubbles [69]. A micro rheometer is used as a tool to measure the linear viscoelastic properties of small volumes of a fluid [19]. For the fuel cell application, a polymeric electrolyte membrane (PEM) is used to produce fuel cell from methanol gas by conducting the protons that are generated by the catalytic oxidation of the gas at the anode [39]. A microbial fuel cell (MFC) can be used as an electricity generator by converting bio-chemical energy to electrical energy [31].

Another important process which involves fluid-particle flow is the targeted drug delivery in cancer therapies. Cancer is one of the most threatening diseases that cause death to human beings. The growth rate of tumor depends on the angiogenesis which is the process in which new blood vessels develop from an existing vasculature through endothelial cell sprouting, proliferation and fusion [64]. Nutrient supplied via new blood vessels results in proliferating of cancer cells, which is in favor of tumor growth. Tumor cells need an adequate blood supply in order to perform vital cellular functions. The modern-day approach to cancer treatment is a multidisciplinary one involving varying combination of surgery, radiation therapy, chemotherapy and targeted therapies. In targeted therapies, a medication or drug is controlled to target a specific pathway in the growth and development of a tumor. Although most of the drugs used to date have proven to be successful on small animals such as mice [30, 86], their efficiency in humans remains highly variable from one patient to another.

Understanding the flow of blood and drug in the capillary bed is very important for investigating the efficiency of drug treatment as they pass from the parent blood vessel to tumor surface via an associated capillary bed. Over the last 15 years, a number of mathematical models for blood vessel formation [12, 47], blood flow and/or particle flow in capillary networks [75, 66, 60, 32] in the area of tumor-induced angiogenesis have been developed. One of these is magnetically targeted drug delivery, which was first introduced by Mosbach and Schroder [57]. This approach involves binding a drug to small biocompatible magnetic particles with
diameters less than 5µm. On the surface, the particles are coated with a surfactant to functionalise some specific tasks. These particles are injected to the blood circulation system and are guided to tumor sites using an external magnetic field. Once targeted to the locations wanted, these particles release drugs to kill the cancer cells on the wall of blood vessel. This treatment has many advantages. Firstly, the size of nanoparticles carrying drugs is small enough to avoid being detected and eliminated from human immune system. The mechanism of magnetic guidance can extend the duration for the particles to be captured to the disease sites. Thus it can release drugs more efficiently.

Driscoll et al. [24] had studied magnetically targeted drug delivery by tracking each individual particle under the influence of Stokes drag force and magnetic force. Grief and Richardson [32] conducted a theoretical analysis of targeted drug delivery using magnetic particles and proposed a two-dimensional network model. In their model, the motion of fluid is described by Poiseuille flow, while the motion of a magnetic particle, due to balancing hydrodynamic and magnetic force, is governed by an advection-diffusion equation for the particle concentration. They found that drug targeting can be achieved by pulling magnetic particles to the edge of the vessel, and that the use of magnetically targeted drug delivery with an externally applied magnetic field is appropriate only for targets close to the surface of the body.

Although intensive research relating to the fluid-particle flow in microchannel and their applications in magnetically targeted drug delivery has been conducted, there are still some aspects that are not fully understood. The use of mathematical models will allow us to gain more understanding of fluid flow, particle flow in fluid, and the interaction between fluid and particles under a magnetic field.

1.2 Objectives

Based on the previous research, in this project, we will study the fluid-particles flow in microchannels under magnetic field. The specific objectives include the
(i) Study the transient behaviour of fluid flow through micro-annulus with boundary slip with particular emphases on the influences of boundary slip on the transient flow rate, velocity and stresses fields.

(ii) Construct a mathematical model and numerical scheme to study the particle-fluid flow in microchannels.

(iii) Construct a mathematical model and numerical scheme to determine the magnetic forces acting on particles by external magnetic sources, and then study the particle motion in fluids under magnetic forces.

1.3 Outline of the Thesis

This thesis consists of six chapters. Chapter one gives the background of the research highlighting the importance of the subject of fluid-particle flow in microchannels. Some specific applications which involve particle-fluid flow are reviewed with particular emphases on the magnetic drug targeting. The objectives are also given in this chapter.

Chapter two reviews relevant theories and previous work closely related to the scope of this study.

Chapter three studies the pressure driven transient flow of a Newtonian fluid in micro-channels. Exact solutions for the velocity and stress fields are derived and the effect of boundary slip is investigated based on the exact solutions.

Chapter four presents a mathematical model and numerical scheme for simulating particle-fluid flows utilizing the finite element method and the arbitrary Lagrangian-Eulerian method. The influence of boundary slip on the velocity and stress fields are also presented.

Chapter five presents the boundary value problem and numerical scheme for the determination of the magnetic vector potential and magnetic forces, followed by the modelling of particle movement in fluids under magnetic forces. A numerical
investigation on the influence of the magnetic field on the particle motion is also presented.

Chapter six presents the conclusions from this study. Further researches are also given in this chapter.
Chapter 2

Literature Review

2.1 General Overview

In this chapter, we review the basic theories and previous work relevant to the study of fluid-particle flow under magnetic forces. In section 2.2, the basic theories for the determination of magnetic fields and magnetic forces are presented. In section 2.3, a review of the fundamental theories for the flow of fluids are given followed by previous work on the development and applications of the subject. In section 2.4, the principal approaches and numerical techniques for simulating the flow of solid particles in fluid is reviewed.

2.2 Modelling of Magnetic Field

Magnetic fields are generated by a magnetic material such as, bar magnets and Earth. The magnetic fields exert forces on electric charges that enter to the field. Unlike electric fields, the magnetic fields only interacts with electric charges that are moving. In addition, the simplest magnetic objects are dipoles.

There are two types of magnetic field: the magnetic induction field $B$ and the magnetic field $H$. The magnetic induction field $B$ (unit of T) is the actual magnetic field. By Faraday’s law, the time variation of the magnetic flux $\Phi_B$ generates the electric field. The magnetic field $H$ is the field that is produced by a wire carrying current. Its strength $H = |H|$ is proportional to the number of turns of that wire.

The magnetic dipole moment $m$ of a magnetic material is defined as the
strength of the magnetic field. The magnetic field is proportional to $|\mathbf{m}|$ and inversely proportional to $r^3$, where $r$ is a distance from the outer of the dipole. The magnetization $\mathbf{M}$ of a magnetic object is defined as the magnetic moment per unit volume, that is,

$$\mathbf{M} = \frac{\mathbf{m}}{V},$$  \hspace{1cm} (2.1)

where $V$ is the volume of the material. This property determines the maximum magnetic by an object that is fully magnetized.

The magnetic force exerted on a dipole $\mathbf{m}$ by a non-uniform static magnetic field $\mathbf{H}$ is given by [74]

$$\mathbf{F} = \nabla (\mathbf{m} \cdot \mathbf{H}).$$ \hspace{1cm} (2.2)

If the current producing the magnetic field does not overlap with the current distribution of the magnetic dipole, we have

$$\nabla \times \mathbf{H} = 0.$$  \hspace{1cm} (2.3)

Recall a vector identity

$$\nabla (\mathbf{a} \cdot \mathbf{b}) = \mathbf{b} \times (\nabla \times \mathbf{a}) + \mathbf{a} \times (\nabla \times \mathbf{b}) + (\mathbf{b} \cdot \nabla)\mathbf{a} + (\mathbf{a} \cdot \nabla)\mathbf{b}.$$ \hspace{1cm} (2.4)

Using (2.3) and (2.4), and assuming that the magnetic dipole is small, the magnetic force in (2.2) becomes

$$\mathbf{F} = (\mathbf{m} \cdot \nabla)\mathbf{H}.$$ \hspace{1cm} (2.5)

In vacuum, the relation between $\mathbf{B}$ and $\mathbf{H}$ is given by

$$\mathbf{B} = \mu_0 \mathbf{H},$$ \hspace{1cm} (2.6)

where $\mu_0 = 4\pi \times 10^{-7} \text{T} \cdot \text{m} \cdot \text{A}^{-1}$ is the permeability of free space.

Inside materials, the three vector fields $\mathbf{B}$, $\mathbf{H}$, and $\mathbf{M}$ are related by the constitutive relation

$$\mathbf{B} = \mu_0 (\mathbf{H} + \mathbf{M}).$$ \hspace{1cm} (2.7)

In magnetostatic problems, a generalized form of the constitutive relation for the magnetic field is given by

$$\mathbf{B} = \mu_0 \mu_r \mathbf{H} + \mathbf{B}_r,$$ \hspace{1cm} (2.8)
where the dimensionless number $\mu_r$ is the relative magnetic permeability, and $B_r$ is the remanent magnetic flux density, which is the magnetic flux density when no magnetic field is present.

Based on the above relations, materials are classified into three categories: paramagnetic, diamagnetic, and ferromagnetic.

In paramagnetic and diamagnetic objects, the magnetic field $B$ is linearly proportional to $H$,

$$B = \mu_0 \mu_r H. \quad (2.9)$$

If $\mu_r > 1$, the material is paramagnetic. If $\mu_r < 1$, the material is diamagnetic. The value $\mu_r$ usually differs from unity by a factor of $10^{-5}$.

For material that are not permanent magnets, $M$ and $H$ are linearly related by

$$M = \chi_m H, \quad (2.10)$$

where $\chi_m$ is the magnetic susceptibility and is dimensionless.

Substituting (2.10) into (2.7), we obtain

$$B = \mu_0 (1 + \chi_m) H. \quad (2.11)$$

Comparing (2.11) and (2.9), we have

$$\mu_r = 1 + \chi_m. \quad (2.12)$$

Thus, for a paramagnetic material, $\chi_m > 0$, and for a diamagnetic material, $\chi_m < 0$.

In general, the governing equations for the electromagnetic field consist of the Maxwell’s equations and the constitutive equations. The Maxwell’s equations are as follows:

$$\nabla \times H = J + \frac{\partial D}{\partial t}, \quad (2.13)$$

$$\nabla \times E = -\frac{\partial B}{\partial t}, \quad (2.14)$$

$$\nabla \cdot B = 0, \quad (2.15)$$

$$\nabla \cdot D = \rho_d, \quad (2.16)$$

8
where $H$ is the magnetic field, $E$ is the electric field, $J$ is the total current density, $D$ is the electric displacement, $\rho_d$ is the free charge density, and $B$ is the magnetic flux density. The constitutive equations are

\begin{align*}
   B & = \mu H, \quad (2.17) \\
   D & = \varepsilon E, \quad (2.18) \\
   J_e & = \sigma (E + \mathbf{v} \times \mathbf{B}) + \rho_d \mathbf{v}, \quad (2.19)
\end{align*}

where $\sigma$ is the electric conductivity, and $\mathbf{v}$ is the velocity. Since metal materials are good conductors, it can be assumed that the field that changes in one part of the system radiates to other part instantaneously. Therefore, $\frac{\partial D}{\partial t}$ can be neglected and (2.13) becomes

$$\nabla \times \mathbf{H} = \mathbf{J}. \quad (2.20)$$

If the magnetic Reynolds number is very small, the term $(\mathbf{v} \times \mathbf{B})$ in (2.19) can be neglected. The Maxwell’s equations together with the constitutive equations can be simplified to the following systems:

\begin{align*}
   \nabla \times \mathbf{H} & = \mathbf{J}, \quad (2.21) \\
   \nabla \times \mathbf{E} & = -\frac{\partial \mathbf{B}}{\partial t}, \quad (2.22) \\
   \nabla \cdot \mathbf{B} & = 0, \quad (2.23) \\
   \nabla \cdot \mathbf{D} & = 0, \quad (2.24) \\
   \mathbf{B} & = \mu \mathbf{H}, \quad (2.25) \\
   J_e & = \sigma \mathbf{E}. \quad (2.26)
\end{align*}

A number of mathematical models and numerical methods for modelling magnetic fields in fluid flows have been carried out. In 2000, Oldenburg et al. [59] studied the flow of ferrofluids in porous media caused by an external magnetic field due to the magnet. To simulate the flow, the external magnetic field, the magnetic field strength and its gradient were calculated by direct equations in two flow characteristics: miscible and immiscible flows. Then the magnetic body force was obtained using the force balance equations solved by the integral finite difference
method. Compared to the experimental results, the two ferrofluid models provide some new simulation capabilities. In 2002, Voltairas et al. [84] developed a general theory based on the ferrohydrodynamics to describe the magnetic drug targeting. They demonstrated that their analytical results, based on the Stokes model, agree with the experimental results in a carotid artery. They derived an upper bound of the mean blood velocity which depends on the external applied magnetic field.

In 2003, Li and Kwok [48] presented a lattice Boltzmann model using a single-relaxation-time approximation as a collision model for studying microfluidic problems which include external pressure and the Lorentz force from external electric and magnetic fields. This model can be potentially applied to magnetic drug targeting. The numerical results are in good agreement with the experimental results in the case of pressure-driven microchannel flows. In 2004, Loukopoudos and Tzirtzilakis [51] derived a mathematical model for biomagnetic fluid dynamics for the flow of the biofluid under an external magnetic field. The numerical results, obtained from the finite difference method, show the effects of a magnetic field on the flow of a biomagnetic fluids including vortex arising and increase in temperature, skin friction coefficient and heat transfer rate with an increase in the magnetic field strength. In 2006, Tzirtzilakis [83] et al. developed a mathematical model, based on the Reynolds averaged Navier-Stokes (RANS) equations, to simulate the turbulent flow of a biomagnetic fluid with a locally applied magnetic field. The low Reynolds number $k - \varepsilon$ turbulence model was used for the turbulence formulation. The computational results show that, when compared between laminar and turbulent flow, the turbulence causes huge reduction on the effect of the magnetic field.

A lot of mathematical models and numerical methods for the motion of particles in fluid flow under an external magnetic field have been developed. In 1984, Driscoll [24] et al studied magnetic targeting of magnetic microspheres carrying drugs to tumorous microvasculature in rat tails. Their calculations of the magnetic force required to move microspheres to blood vessel walls, the fluid drag force, and the interparticle force were used for the analysis of microsphere capture from the experiment. In 2005, Rotariu and Strachan [65] investigated magnetic targeting of
drugs in tumor capillaries deep inside the human body using magnetic particles and an external needle magnet. A two dimensional finite element model based on the FEMM software using the magnetohydrodynamics and the motion of particles was used to solve the problem. The numerical results show that the particles of size 1 µm can be captured at a distance up to 15 cm.

Aviles et al. [7] presented a two dimensional Newtonian model with magnetic forces and used FEM to simulate the movement of magnetic drug carrier particles used to treat carotid artery diseases under the affect of an external magnetic field. In addition to a permanent magnet, the use of a near-skin implanted ferromagnetic wire was shown to improve the collection efficiency of the magnetic particles. Ally et al. [4] used the finite element package ANSYS to study magnetically targeted aerosols in lung cancer. They computed drag forces using the Stokes flow and the magnetic forces were used to calculate the motion of particles by Newton’s second law. The experimental results and numerical results are in good agreement and show that the magnetic field gradient and aerosol particle concentration are important factors for magnetic targeting of aerosols.

Chen et al. [18] developed a two dimensional mathematical model using the Navier-Stokes equations and Maxwell’s equations, to simulate magnetic targeting of magnetic particles using a magnetizable intravascular stent (MIS) inserted intro a blood vessel. They suggested that the radii of the MIS and its wire are important parameters for the collection efficiency. Grief and Richardson [32] conducted a theoretical analysis of the targeted drug delivery using magnetic particles and proposed a two-dimensional network model. In their model, the motion of fluid is described by Poiseuille flow, while the motion of the magnetic particles, due to balancing hydrodynamic and magnetic force, is governed by an advection-diffusion equation for the particle concentration. They found that drug targeting can be achieved by pulling magnetic particles to the edge of the vessel, and that the use of magnetically targeted drug delivery with an externally applied magnetic field is appropriate only for targets close to the surface of the body.

by using ferromagnetic particles as seeds to collect magnetic drug carrier particles at a specific site in the body. They used the Navier-Stokes equations to model the fluid flow, and the Maxwell equations to capture the magnetic potentials, and the force balance equation to calculate the motion of the carrier particles. The magnetic particle-particle interactions were neglected. The numerical results show that using the ferromagnetic seeds increases the performance of the magnetic drug targeting determined by the capture cross section.

Furlani and Furlani [27] developed a mathematical model based on the magnetic force, the drag force under the Stokes flow and the motion of magnetic particles for magnetic drug delivery in microvessels. They derived analytical expressions for the prediction of particle trajectory and the volume fraction of magnetic particles required to ensure capture of the particle, which is conversely proportional to the square of the radius of particles. Chertok et al. [20] used a simple mathematical model for the magnetohydrodynamic problem to distinguish the tumor brain from healthy brain by using magnetic nanoparticles and an external magnet. The prediction results were confirmed by the in vivo ratio of nanoparticle capture in bearing rats. In 2008, Chen [17] et al. developed a high-gradient magnetic separator to improve magnetic particle separation from blood flow in magnetic targeting application by introducing an array of capillary tubing and ferromagnetic wires which was immersed in an external magnetic field from two permanent magnets. They used a three dimensional finite element model, based on the equations similar to those in [18], to investigate the effect of the tubing-wire configurations. The numerical results show that an optimal design is to have bi-directionally alternating wires and tubes.

2.3 Modelling of Fluid Flow

Fluid dynamics involves the study of fluids in motion. More specifically, it concerns the kinematics of the flow field, and the stress distribution throughout the field. Based on the compressibility, fluids can be classified into compressible fluids and
incompressible fluids. Based on the constitutive equations, fluids can be categorized into three types: non-viscous, Newtonian, and non-Newtonian.

For most problems, liquids can be treated as incompressible fluids, and in general, gasses except low speed gas flows must be considered as compressible fluids. Furthermore, many common fluids, such as air and water, can be modelled as Newtonian fluids.

The governing equations of the flow of an incompressible Newtonian fluid consists of the equations of motion

$$\frac{\partial \sigma_{ij}}{\partial x_j} + \rho_f X_i = \rho_f \frac{Du_i}{Dt},$$  \hspace{1cm} (2.27)

the equation of continuity

$$\text{div}(u) = \frac{\partial u_j}{\partial x_j} = 0,$$  \hspace{1cm} (2.28)

and the constitutive equations

$$\sigma_{ij} = -p\delta_{ij} + 2\mu d_{ij},$$  \hspace{1cm} (2.29)

where $\frac{Du_i}{Dt} = \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j}$ is the material derivative, $\sigma_{ij}$ are the components of stress tensor, $\rho_f$ is the density of fluid, $X_i$ are the components of body force in the $x_i$ direction, $\delta_{ij}$ is the Kronecker symbol, $u_i$ are the velocity of fluids in the $x_i$ direction, $p$ is the pressure, $\mu$ is the viscosity of fluid, and $d_{ij}$ are the components of the rate of deformation (also known as strain rate) and is related to the velocity by

$$d_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right).$$

The above equations are all field equations which must be satisfied at all points within a continuum. To solve the above equations, substituting (2.33) into (2.27), we obtain

$$\rho_f \frac{Du_i}{Dt} = \rho_f X_i - \frac{\partial p}{\partial x_j} \delta_{ij} + \mu \left( \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial^2 u_i}{\partial x_i \partial x_j} \right).$$  \hspace{1cm} (2.30)

Using the continuity equation (2.28), we have

$$\frac{\partial^2 u_i}{\partial x_i \partial x_j} = \frac{\partial}{\partial x_i} \left( \frac{\partial u_j}{\partial x_j} \right) = 0,$$
and thus equation (2.30) becomes

\[
\frac{D u_i}{Dt} = X_i - \frac{1}{\rho_f} \frac{\partial p}{\partial x_i} + \frac{\mu}{\rho_f} \nabla^2 u_i, \tag{2.31}
\]

which are the so-called Navier-Stokes equations for incompressible Newtonian fluids. It should be noted that the Navier-Stokes equations for other kinds of fluids can be derived using the same process but with different constitutive equations. The Navier-Stokes equations (2.31) together with the continuity equation (2.28) constitute a system of four partial differential equations for unknown variables \( u, v, w, \) and \( p \) and thus are solvable in principle. These four partial differential equations define all possible motions of an incompressible Newtonian fluid. The feature which distinguishes one flow situation from another is the nature of the boundary conditions satisfied by the velocity field \( u \) and \( p \).

In historical development of fluid theories and simulations, many studies have shown that fluids behave approximately like Newtonian fluid for which the shear stress in fluid is linearly proportional to the deformation rate, namely

\[
\mu = \text{constant}. \tag{2.32}
\]

However, most fluids are non-Newtonian, that is, the shear stress is related to the deformation rate nonlinearly by

\[
\sigma_{ij} = -p\delta_{ij} + 2\mu(\dot{\gamma})d_{ij}, \tag{2.33}
\]

where \( \dot{\gamma} \) is the shear rate. Many non-Newtonian models for fluids have been suggested. Each model can be used to describe different fluid behaviour and contains different parameters. For bloods, a summary of non-Newtonian models of blood viscosity including the model parameters is given in [42], as in Table 2.1.

Many studies have been carried out to study the non-Newtonian behaviour of fluid and many models have been developed including, for example, the Carreau model, the Power-law model, the Casson model, and the generalized Power-law model. Mooney and Black [56] investigated the flow of raw rubber in extrusion operations. Both two and three dimensional flows were investigated. It was concluded that the non-Newtonian model was in a good agreement with Nadai’s law of
Table 2.1: Blood viscosity models, $\mu$, given in Poise (P) as a function of strain rate, $\dot{\gamma}$, given in $s^{-1}$ [42].

<table>
<thead>
<tr>
<th>Blood model</th>
<th>Effective viscosity $\mu$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Newtonian</td>
<td>$\mu = 0.0345$ P</td>
</tr>
</tbody>
</table>
| Carreau             | $\mu = \mu_\infty + (\mu_0 - \mu_\infty)[1 + (\lambda \dot{\gamma})^2]^{(n-1)/2}$  
                        | where $\mu_0 = 0.56$ P, $\mu_\infty = 0.0345$ P,  
                        | $\lambda = 3.313$ s and $n = 0.3568$                     |
| Walburn-Schneck     | $\mu = C_1 e^{C_2 H}[e^{C_4 (TPMA/H^2)}](\dot{\gamma})^{-C_3 H}$,  
                        | where $C_1 = 0.00797$, $C_2 = 0.0608$,  
                        | $C_3 = 0.00499$, $C_4 = 14.585 l g^{-1}$,  
                        | $H = 40\%$ and $TPMA = 25.9 gl^{-1}$                    |
| Power Law           | $\mu = \mu_0(\dot{\gamma})^{n-1}$                            
                        | where $\mu_0 = 0.035$ and $n = 0.6$                       |
| Casson              | $\mu = [(\eta^2 J_2)^{1/4} + 2^{-1/2} \tau_y^{1/2}]^2 J_2^{-1/2}$,  
                        | where $|\dot{\gamma}| = 2 \sqrt{J_2}$, $\tau_y = 0.1(0.625 H)^3$,  
                        | $\eta = \eta_0(1 - H)^{-2.5}$ with $\eta_0 = 0.012$ P and $H = 0.37$ |
| Generalised Power Law | $\mu = \lambda |\dot{\gamma}|^{n-1}$,                            
                          | $\lambda(\dot{\gamma}) = \mu_\infty + \Delta \mu \exp \left[-(1 + \frac{|\dot{\gamma}|}{a}) \exp \left(\frac{-b}{|\dot{\gamma}|}\right)\right]$,  
                          | $n(\dot{\gamma}) = n_\infty - \Delta n \exp \left[-(1 + \frac{|\dot{\gamma}|}{c}) \exp \left(\frac{-d}{|\dot{\gamma}|}\right)\right]$,  
                          | where $\mu_\infty = 0.035$, $n_\infty = 1.0$, $\Delta \mu = 0.25$,  
                          | $\Delta n = 0.45$, $a = 50$, $b = 3$, $c = 50$ and $d = 4$ |
steady creep. Ali and Hayat [3] employed the Carreau model to study the peristaltic mechanism of a non-Newtonian fluid in an asymmetric channel. A perturbation technique was used to derive solutions in terms of power of a small Weissenberg number. Batra and Jena [8] studied the flow of a non-Newtonian fluid through a curved blood vessel using the Casson model. It was concluded that the ratio of the friction factor for the fluid in a curved tube to that in a straight tube depends on the Dean number and yield number.

Buick [11] studied the flow of a non-Newtonian fluid in a single-screw extruder using the power-law model. The lattice-Boltzmann method was used to simulate the behaviour of fluid flow. It was reported that the non-Newtonian model has an effect on the behaviour of the flow. Mandal et al. [52] investigated the flow of a non-Newtonian fluid through a simplex atomizer by simulating the transient flow of a viscous power-law fluid in the simplex atomizer using the volume-of-fluid method. It is found that the varying of power law index has no influence on the problem parameters including the angle and discharge coefficient. Mangadoddy et al. [53] investigated the flow of an incompressible non-Newtonian fluid in crossing over a bank of circular cylinders by using a power-law model. It is found that the Nusselt number depends on the Reynolds number, Prandtl number or Peclet number, power law index and the voidage of tube banks. Ismail et al. [41] studied blood flow in a tapered stenotic artery using the generalised power-law model. A finite difference scheme was used to simulate the behaviour of blood flow. They reported that the model showed lower values of the axial velocity profiles, flow rate, and wall shear stress, and higher values of the resistive impedances, compared with the Newtonian model. Soh and Mureithi [73] studied the flow of a non-Newtonian fluid down a heated inclined plane using the power-law temperature-dependent viscosity. The exact solutions were obtained using Bessel functions. The numerical solutions showed that the Froude number had significant effects on the flow velocity and temperature.

Gao and Wang [29] theoretically presented a mathematical model to study the incompressible non-Newtonian fluid using the generalized power-law model. A
similarity solution was detained using a perturbation technique and the Schander’s fixed point theorem. The asymptotic behaviour of the solutions was also investigated.

2.4 Modelling of Particle-fluid Flow

Here we consider the cases where fluids adhere to rigid, but possible moving, surfaces bounding the fluids. It is evident that on a rigid surface, the normal component of the fluid velocity must be the same as that of the rigid surfaces, as fluids cannot penetrate the solid. For the tangential component of fluid velocity, two different boundary conditions may be used: no-slip condition and slip condition. For the no-slip condition, it is assumed that the tangential velocity component is likewise the same as that of the rigid surface. For the slip condition, it is assumed that slip can occur between the fluid and the rigid body. It has been found that the no-slip condition accords with experimental observation on most real materials.

Various mathematical models and numerical algorithms for the motion of particles in fluids have been developed. The numerical methods can be divided into three types including the continuum theory, the Lagrangian numerical simulation (LNS), and the direct numerical simulation (DNS). Based on these approaches, many research has been done to simulate the behaviour of particle flow in fluid.

The DNS approach is able to fully couple the motion of fluid flow and particles. In this method, various numerical methods have been developed. The first method is the Arbitrary Lagrangian-Eulerian method (ALE). Takashi and Hughes [78] studied the vibration of a circular cylinder in a circular domain filled with a viscous fluid. In the ALE method, the material time derivative of a physical property $\varphi$ is given by

$$\dot{\varphi} = \varphi' + c_i \varphi_{,i},$$  \hspace{1cm} (2.34)

where $\varphi'$ is the referential time derivative keeping coordinates in the referential domain constant, and $c_i$ is the convective velocity given by

$$c_i = u_i - \ddot{u}_i,$$ \hspace{1cm} (2.35)
where \( u_i \) is the material velocity and \( \hat{u}_i \) is the mesh velocity. Applying the convective velocity to the equations of fluid motion, the ALE description of the Navier-Stokes equations and the continuity equation is given by

\[
\rho u'_i + \rho(u_j - \hat{u}_j) \frac{\partial u_j}{\partial x_j} = \frac{\partial \tau_{ij}}{\partial x_j} + f_i, \quad \text{in } \Omega_F(t),
\]

\[
\frac{\partial u_i}{\partial x_i} = 0, \quad \text{in } \Omega_F(t),
\]

(2.36)

where \( u_i \) is the velocity vector of the fluid, \( \rho \) is the density, \( \tau_{ij} \) is the stress tensor, and \( f_i \) is the body force vector. The fluid motion is described in the moving spatial domain \( \Omega_F(t) \), while the domain occupied by the moving rigid body is denoted by \( \Omega_G(t) \). The interface \( \Gamma_I(t) \) between \( \Omega_F(t) \) and \( \Omega_G(t) \) moves as the domain \( \Omega_G(t) \) changes its position. The boundary condition consists of two parts given by

\[
\begin{align*}
  u_i &= g_i \text{ on } \Gamma_g, \\
  t_i &= \tau_{ij} n_j = h_i \text{ on } \Gamma_h,
\end{align*}
\]

(2.37)

where \( t_i \) is the traction, \( n_i \) is the unit outward normal vector to \( \Gamma_h \), and both \( g_i \) and \( h_i \) are specified functions of space and time. With the no-slip condition on the moving interface \( \Gamma_I(t) \), the unknown velocity \( u^I_i \) on \( \Gamma_I(t) \) can be determined by

\[
u^I_i = \hat{u}_i.
\]

(2.38)

Based on this ALE formulation, Hu [36] simulated the Poiseuille flow of solid-fluid mixtures of 400 solid particles in fluid contained in a vertical channel.

The second method in DNS is the lattice-Boltmann method (BLM). Ladd [44] developed the method to solve the Navier-Stokes equations and gave some simple examples in solving the Stokes equations. In this method, the mass density \( \rho \), the momentum density \( j = \rho u \), and the momentum flux \( \Pi \) are given by

\[
\begin{align*}
  \rho &= \sum_i n_i, \\
  j &= \sum_i n_i c_i, \\
  \Pi &= \sum_i n_i c_i c_i,
\end{align*}
\]

(2.39)

where \( n_i(r, t) \) is the continuous velocity distribution function of the discretized quantities \( r, t, \) and \( c_i \), where \( r \) is the number of particles at a particular node of the lattice at time \( t \) with velocity \( c_i \).
In 2000, Pereira [62] studied the fluid-particle flow problem using the finite difference model. The motion of spherical and cylindrical particles in steady state Newtonian and non-Newtonian fluid flow inside a tube was investigated by determining the ratio between the additional force and the drag force on the particle, and the ratio between the additional force on the duct wall and the drag force on the particles. In 2004, Longest et al. [50] presented a mathematical model based on the Eulerian-Lagrangian method to simulate the trajectories of discrete blood particles including particle-wall interactions and the quantification of near-wall stasis, using the near-wall residence time model. They used the CFX finite volume package to apply a femoral bypass end-to-side anastomosis as a real model geometry. The numerical results show that an extended form of the particle trajectory equation is needed when particle-wall interaction terms dominate.

In 2006, Bikard et al. [9] studied a three dimensional mathematical model for the fluid flow around a rigid sphere suspended in a Newtonian matrix and submitted to a simple shear. They used the Rem3D finite element software to calculate the flow and the hydrodynamic stresses around the particle. The numerical results agree with the results from the direct calculation, and the matrix has significant effects on the proximity of the matrix walls toward the particles. In 2007, Al Quddus et al. [1] developed a mathematical model based the Navier-Stokes equations and an arbitrary Lagrangian Eulerian method and used the finite element approach to study the motion of particles in fluids in infinite and finite length channels. The simulation results are in good agreement with other analytical and numerical results for the Stokes hindrance factors in capillary flows. A double exponential relation between the wall correction factors and particle to channel radii ratio was obtained. They also investigated the effect of entrance and exit of an open channel, and a capped end of a channel.
2.5 Concluding Remarks

A large number of studies related to the fluid flow and particle flow in microscale have been carried out over the last few decades in order to understand the interaction between fluids and particles. Many mathematical models have been used to simulate the fluid-particle flow. A large amount of work have also been conducted to study the effect of magnetic fields to fluid flow. This research will focus on the fluid-particle flow through microchannels under magnetic fields.
Chapter 3

Transient Flow of Fluids through Micro-annuals

3.1 General Overview

From the fundamental principles of continuum mechanics, the flow of incompressible Newtonian fluids is governed by the continuity equation, the Navier-Stokes equations and a set of boundary conditions. Traditionally the so-called no-slip boundary condition is used, namely the fluid velocity relative to the solid is assumed to be zero on the fluid-solid interface [71]. However, evidences of slip of a fluid on a solid surface have been reported [63]. Chauveteau [16], Tuinier and Taniguchi [82], and Vargas and Manero [22] studied the flow of polymer solutions in porous media and showed that the apparent viscosity of the fluids near the wall is lower than that in the bulk and consequently the fluids can exhibit the phenomenon of apparent slip on the wall. More recently, experiments in micrometer scale and molecular dynamic simulations showed that the flow of fluids in microsystems is granular and slip can occur on the fluid-solid interface [14, 15, 77, 91, 95]. Hence, under certain conditions such as those investigated in [81, 85], the no-slip condition is not acceptable for fluid flows in microchannels. On the other hand, many experimental results have provided evidences to support the Navier slip condition [38, 63, 93], namely the fluid velocity component tangential to the solid surface, relative to the solid surface, is proportional to the shear stress on the fluid-solid in-
terface. The proportionality is called the slip length which describes the _slipperiness_ of the surface \[45, 93\]. Some attempts have also been made to derive alternative formulae for the determination of the slip length \[79\], and to use nanotechnologies for the surface treatment of microchannels so as to achieve large slip for maximizing the transport efficiency of fluids through microchannels. Based on the nature of boundary slip of microflows, we will use the Navier slip condition for fluid flow in micro-annuals.

Over the last couple of decades, many investigations have been made to study various flow problems of Newtonian and non-Newtonian fluids with the no-slip boundary condition or a slip boundary condition \[13, 21, 23, 46, 54, 61, 67, 68, 81, 94\]. Although exact and numerical solutions to many flow problems of Newtonian fluids under the no-slip assumption have been obtained and are available in literature \[71, 87, 88, 89\], very few exact solutions for the slip case are available in literature. Recently, some steady state slip solutions for the flows through a pipe, a channel and an annulus have been obtained \[55, 92\]. An exact solution for the transient flow through microtubes has also been derived and discussed in the paper \[90\].

Motivated by the previous work, we study the transient flow of an incompressible Newtonian liquid through a micro-annual with a slip boundary in this paper. The work is basically an extension of our previous work in \[90\]. Here we should address that micro-devices with annulus geometry, such as microreactor with multicylindrical mixer structure \[34\], are extremely difficult to fabricate using current microfabrication technology. The rest of the paper is organized as follows. In the following section, we first define the problem and then present its mathematical formulation. In section 3, we solve the underlying boundary value problem to derive the exact solution for the velocity field and show that the solution includes some existing known solutions as special cases. In section 4, we derive exact solutions for the flow rate, the rate of deformation tensor and the stress field in the fluid using the exact solution of velocity field derived in section 3. In section 5, an analysis is carried out to study the influence of the slip parameter on the flow behaviour. Finally a conclusion is given in section 6.
3.2 Governing Equations

Consider the transient flow of an incompressible Newtonian liquid through a circular annular of inner radius \( a \) and outer radius \( R \) with the \( z \)-axis being in the axial direction as shown in Figure 3.1. We limit our analysis to fully developed flow and assume that the slip length does not change along the flow. The field equations governing the flow include the continuity equation and the Navier-Stokes equations. As the flow is axially symmetric and fully developed, there is no swirling flow and the velocity components in the radial and transverse directions vanish, namely \( \mathbf{v} = (v_r, v_\theta, v_z) = (0, 0, u) \). Thus, from the continuity equation and the Navier-Stokes equations, as shown in [90], \( u \) must satisfy the following equation

\[
\frac{\mu}{\rho} \left( \frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} \right) - \frac{\partial u}{\partial t} = \frac{1}{\rho} \frac{\partial p}{\partial z} .
\] (3.1)

Since a wide range of functions can be expressed in terms of Fourier series, in this work, we consider the fluid flow driven by the pressure field with a pressure gradient that can be expressed by the Fourier series

\[
\frac{\partial p}{\partial z} = a_0 + \sum_{n=1}^{\infty} \left[ a_n \cos(n\omega t) + b_n \sin(n\omega t) \right] := q(t).
\] (3.2)

To completely define the problem, the field equations must be supplemented by the boundary condition. In this work, we use the Navier slip boundary condition. That is, on the solid-fluid interfaces \( r = a \) (inner surface) and \( r = R \) (outer surface), the axial fluid velocity, relative to the solid surface, is proportional to the shear stress
on the interface. For Newtonian fluids, the shear stress is related to the shear strain rate by
\[ \sigma_{rz} = \mu \frac{\partial u}{\partial z} \]
where \( \mu \) is the fluid viscosity. Thus, for the case where the rigid micro-annulus is fixed spatially, the Navier slip condition can be written in the form of
\[ u(a, t) = \pm l_1 \frac{\partial u}{\partial r}(a, t), \quad u(R, t) = \pm l_2 \frac{\partial u}{\partial r}(R, t), \]  
(3.3)
where \( l_1 \) and \( l_2 \) denote the slip parameters of the inner surface and the outer surface respectively. In this study, we assume that the slip parameter does not change along the flow. The signs for the terms on the right hand sides of the above equations have been discussed by various authors. In literature, all the four possible cases are considered and the physically feasible cases are determined based on the solution derived. Here, we give a different method for choosing the sign for the terms on the right hand sides of the equations in (3.3) without the need of finding the solution first, as detailed below.

From the physics of fluids, when the fluid moves relative to the solid surface in the tangential direction of the solid surface, the relative movement of the fluid particles will be restricted by a resistance force acting on the opposite direction of the relative movement. Let the unit outward normal vector of the surface \( S \) of the fluid is \( \mathbf{n} = (n_1, n_2, n_3) \), and the positive tangential direction is \( \mathbf{t} = (t_1, t_2, t_3) \). Suppose the stress tensor in the fluid is \( \sigma_{ij} \), then the surface traction on \( S \) is \( X_i = \sigma_{ij}n_j \) which has the tangential component \( f_i = X_it_i = \sigma_{ij}n_jt_i \), where we have used the index notation with the repeated literal indexes representing summation over the index range. On the other hand, the velocity component of the fluid relative to the solid surface on the tangential direction is \( v_t - v_{st} = (v_i - v_{si})t_i \). Hence the Navier type boundary condition for Newtonian fluids can be written as
\[ v_t - v_{st} = -\frac{l_f}{\mu} \quad \text{or} \quad (v_i - v_{si})t_i = -\frac{l_i(n_i n_j t_j)}{\mu}. \]  
(3.4)

The negative sign in the above equations is to indicate that the direction of the relative tangential velocity is opposite to the surface traction force exerted on the fluid by the solid surface. Now for our problem in the \((r, \theta, z)\) system, \( \mathbf{v} = (0, 0, u) \), \( \mathbf{v}_s = (0, 0, 0) \). For the outer surface \( r = R \), \( \mathbf{n} = (1, 0, 0) \) and \( \mathbf{t} = \)
\((0, 0, 1)\), and so \(v_t - v_{st} = v_i r_i = u\) and \(f_t = \sigma_{rz} = \mu \frac{\partial u}{\partial r}\) and consequently \((3.3)_2\) for the outer surface takes the following form
\[
u(R, t) = -l_2 \frac{\partial u}{\partial r}(R, t).
\]

For the inner surface \(r = a, n = (-1, 0, 0)\) and \(t = (0, 0, 1)\), and so \(v_t - v_{st} = v_i r_i = u\) and \(f_t = -\sigma_{rz} = -\mu \frac{\partial u}{\partial r}\) and consequently \((3.3)_1\) for the inner surface takes the following form
\[
u(a, t) = l_1 \frac{\partial u}{\partial r}(a, t).
\]

It should also be addressed here that for \(l_i = 0\), conditions \((3.5)-(3.6)\) reduce to the no-slip boundary condition; while, for \(l_i \to \infty\), equations \((3.5)\) and \((3.6)\) give the surface traction condition for perfectly smooth surfaces, i.e., \(\sigma_{rz}(a, t) = \sigma_{rz}(R, t) = 0\).

**Remark 2.1** The formula \((3.4)\) is more precise than \((3.3)\) and is more suitable for application in numerical analysis.

### 3.3 Exact Solution for Velocity Field

To solve equation \((3.1)\), firstly we use complex number to express the above Fourier series by exponential functions, namely
\[
\frac{\partial p}{\partial z} = \text{Re}\left(\sum_{n=0}^{\infty} c_n e^{in\omega t}\right),
\]
where \(c_n = a_n - b_n i\) and \(e^{in\omega t} = \cos(n \omega t) + i \sin(n \omega t)\).

From the linear property of equation \((3.1)\), we have \(u = \sum_{n=0}^{\infty} \text{Re}(u_n)\), where \(u_n\) is defined by
\[
\frac{\mu}{\rho} \left(\frac{\partial^2 u_n}{\partial r^2} + \frac{1}{r} \frac{\partial u_n}{\partial r}\right) - \frac{\partial u_n}{\partial t} = \frac{c_n}{\rho} e^{in\omega t}.
\]

As in [90], let
\[
u_n = f_n(r) e^{in\omega t}.
\]
Then, we have
\[
\frac{\mu}{\rho} \left( \frac{\partial^2 f_n}{\partial r^2} + \frac{1}{r} \frac{\partial f_n}{\partial r} \right) - in\omega f_n = \frac{c_n}{\rho},
\] (3.10)

For \( n = 0 \), equation (3.10) has the following general solution
\[
f_0(r) = (A_1 + A_2lnr) + \frac{c_0}{4\mu} r^2.
\] (3.11)

For \( n \geq 1 \), equation (3.10) can be written as
\[
\hat{r}^2 \frac{\partial^2 f_n}{\partial \hat{r}^2} + \frac{\partial f_n}{\partial \hat{r}} + f_n = \frac{c_n}{\beta_n^2 \mu} \hat{r}^2,
\] (3.12)
where \( \beta_n^2 = n \beta^2 \) in which \( \beta^2 = -\frac{\omega}{\mu} i \), and \( \hat{r} = \beta_n r \).

As the associated homogeneous equation is the zero-order Bessel equation, equation (3.12) has the following general solution
\[
f_n = d_n J_0(\hat{r}) + e_n Y_0(\hat{r}) + \frac{c_n}{\beta_n^2 \mu} = d_n J_0(\beta_n r) + e_n Y_0(\beta_n r) + \frac{c_n i}{\rho \omega},
\] (3.13)
where \( d_n \) and \( e_n \) are integration constants; \( J_0 \) and \( Y_0 \) denote the zero-order Bessel functions of the first kind and the second kind respectively. Thus, we have
\[
u = \sum_{n=0}^{\infty} \Re(\nu_n) = A_1 + A_2 ln(r) + \frac{a_0}{4\mu} r^2
\]
\[+ \sum_{n=1}^{\infty} \Re \left[ \left( d_n J_0(\beta_n r) + e_n Y_0(\beta_n r) + \frac{c_n i}{\rho \omega} \right) e^{-i\omega t} \right],
\] (3.14)
from which we obtain
\[
\frac{\partial \nu}{\partial r} = A_2 \frac{1}{r} + \frac{a_0}{2\mu} r - \Re \sum_{n=1}^{\infty} [d_n J_1(\beta_n r) + e_n Y_1(\beta_n r)] \beta_n e^{-i\omega t},
\] (3.15)
where, in the above formulation, we have used the identities
\[
\frac{dJ_0(x)}{dx} = -J_1(x), \quad \frac{dY_0(x)}{dx} = -Y_1(x).
\] (3.16)

Substituting (3.14) and (3.15) into boundary conditions (3.5-3.6) yields
\[
\left( A_1 + A_2 ln(a) + \frac{a_0}{4\mu} a^2 - A_2 \frac{1}{2} l_1 - \frac{a_0}{2\mu} a l_1 \right) + \Re \sum_{n=1}^{\infty} [d_n J_0(\beta_n a) + e_n Y_0(\beta_n a)]
\]
\[+ \frac{c_n i}{\rho \omega} l_1 \beta_n d_n J_1(\beta_n a) + l_1 \beta_n e_n Y_1(\beta_n a)] e^{-i\omega t} = 0,
\]
\[
\left( A_1 + A_2 ln(R) + \frac{a_0}{4\mu} R^2 + A_2 \frac{1}{2} l_2 + \frac{a_0}{2\mu} R l_2 \right) + \Re \sum_{n=1}^{\infty} [d_n J_0(\beta_n R) + e_n Y_0(\beta_n R)]
\]
\[+ \frac{c_n i}{\rho \omega} l_2 \beta_n d_n J_1(\beta_n R) - l_2 \beta_n e_n Y_1(\beta_n R)] e^{-i\omega t} = 0.
\] (3.17)
For the above equations to hold for any instant of time \( t \), we require that
\[
A_1 + A_2 \left( \ln(a) - \frac{1}{4}l_1 \right) = -\frac{a_0}{4\mu} (a^2 - 2al_1),
\]
\[
A_1 + A_2 \left( \ln(R) + \frac{1}{R}l_2 \right) = -\frac{a_0}{4\mu} (R^2 + 2Rl_2),
\]
\[
d_n [J_0(\beta_n a) + l_1 \beta_n J_1(\beta_n a)] + e_n [Y_0(\beta_n a) + l_1 \beta_n Y_1(\beta_n a)] = -\frac{a_0}{\rho \omega}, \tag{3.18}
\]
\[
d_n [J_0(\beta_n R) - l_2 \beta_n J_1(\beta_n R)] + e_n [Y_0(\beta_n R) - l_2 \beta_n Y_1(\beta_n R)] = -\frac{a_0}{\rho \omega}.
\]

Solving the above system of equations for \( A_1, A_2, d_n \) and \( e_n \), and then submitting them into (3.14), we obtain
\[
u = -\frac{a_0 R^2}{4\mu} \left( 1 - \left( \frac{r}{R} \right)^2 + \frac{2l_2}{R} + \frac{1}{\ln \left( \frac{R}{r} \right)} \frac{(\ln \frac{R}{r} - \ln \frac{r}{R})}{\ln \left( \frac{R}{r} \right)} \right)
- \Re \sum_{n=1}^{\infty} \frac{c_n e^{i\nu t \omega}}{\rho \omega} \left\{ \frac{N_1}{D} J_0(\beta_n r) + \frac{N_2}{D} Y_0(\beta_n r) - 1 \right\}, \tag{3.19}
\]
where
\[
N_1 = N_1(a, R, \beta_n, l_1, l_2) = Y_0(\beta_n R) - Y_0(\beta_n a) - l_2 Y_1(\beta_n R) + l_1 Y_1(\beta_n a),
\]
\[
N_2 = N_2(a, R, \beta_n, l_1, l_2) = J_0(\beta_n a) - J_0(\beta_n R) + l_2 J_1(\beta_n a) + l_1 J_1(\beta_n R),
\]
\[
D = D(a, R, \beta_n, l_1, l_2) = J_0(\beta_n a) Y_0(\beta_n R) - J_0(\beta_n R) Y_0(\beta_n a)
+ [J_1(\beta_n a) Y_0(\beta_n R) - J_0(\beta_n R) Y_1(\beta_n a)] l_1 \beta_n
+ [J_1(\beta_n R) Y_0(\beta_n a) - J_0(\beta_n a) Y_1(\beta_n R)] l_2 \beta_n
+ [J_1(\beta_n R) Y_1(\beta_n a) - J_1(\beta_n a) Y_1(\beta_n R)] l_1 l_2 \beta_n^2. \tag{3.20}
\]

**Remark 3.1** If \( l_1 = l_2 = 0 \), solution (3.19) becomes
\[
u = -\frac{a_0 R^2}{4\mu} \left\{ 1 - \left( \frac{r}{R} \right)^2 + \left[ 1 - \left( \frac{a}{R} \right)^2 \right] \ln \left( \frac{r}{R} \right) \right\}
- \Re \sum_{n=1}^{\infty} \frac{c_n e^{i\nu t \omega}}{\rho \omega} \left\{ \frac{[Y_0(\beta_n R) - Y_0(\beta_n a)] J_0(\beta_n r) + [J_0(\beta_n a) - J_0(\beta_n R)] Y_0(\beta_n r)}{J_0(\beta_n a) Y_0(\beta_n R) - J_0(\beta_n R) Y_0(\beta_n a)} - 1 \right\}, \tag{3.21}
\]
which is the solution for the traditional no-slip case [40].

**Remark 3.2** If \( a_0 = -A \in \mathbb{R}, c_n = 0 \) for all \( n \geq 1, l_1/R = l_2/R = l, a = \kappa R, u = -v_z a_0 R^2/4\mu \) then the solution reduces to a recent result given in equation (3.19) in reference [55].

### 3.4 Exact Solution of the Flow Rate and Stress Field

From the axial velocity solution (3.19), the flow rate can be determined as
\[
Q(t) = \int_a^R 2\pi r u(r, t) \, dr = Q_0 + \sum_{n=1}^{\infty} Q_n, \tag{3.22}
\]
where $Q_0$ and $Q_n$ are respectively the flow rate corresponding to the constant component and the $n$th harmonic component of the pressure gradient and

$$Q_0 = -\frac{a_0 R^4}{8l^2} \left\{ \left[ 1 - \left( \frac{a}{R} \right)^2 \right] \left[ 1 - \left( \frac{a}{R} \right)^2 + \frac{4l^2}{R^2} \right] - \frac{1}{\ln \left( \frac{a}{R} + \frac{l}{R} \right) + \frac{a}{R}} \left[ 1 - \left( \frac{a}{R} \right)^2 + \frac{2l^2}{R^2} - \frac{2l^2}{R^2} \ln \frac{a}{R} \right] \right\},$$

$$Q_n = -Re \left\{ \left[ \frac{N_1}{R} \int_a^R r J_0(\beta_n r) \, dr + \frac{N_2}{D} \int_a^R r Y_0(\beta_n r) \, dr - \frac{1}{2} \left( R^2 - a^2 \right) \right] \frac{2\pi c_n \omega}{n \rho \omega} \right\}.$$  

From the identities

$$\frac{d}{dx} [x J_1(x)] = x J_0(x), \quad \frac{d}{dx} [x Y_1(x)] = x Y_0(x),$$

we have

$$\int_a^R r J_0(\beta_n r) \, dr = \frac{1}{\beta_n} [R J_1(\beta_n R) - a J_1(\beta_n a)],$$

$$\int_a^R r Y_0(\beta_n r) \, dr = \frac{1}{\beta_n} [R Y_1(\beta_n R) - a Y_1(\beta_n a)].$$

Thus, by substituting the above formula into (3.22), we have

$$Q_n = -Re \left\{ \left[ \frac{N_1}{\beta_n D} [R J_1(\beta_n R) - a J_1(\beta_n a)] + \frac{N_2}{\beta_n D} [R Y_1(\beta_n R) - a Y_1(\beta_n a)] - \frac{1}{2} \left( R^2 - a^2 \right) \right] \frac{2\pi c_n \omega}{n \rho \omega} \right\}.$$  

Remark 4.1 From the above solution form, it is not immediately clear whether the transient flow rate increases as $l_1$ and/or $l_2$ increases, and thus we will study this in section 5.

The stress in the fluid is related to the velocity field by the following constitutive equations

$$\sigma = -pI + 2\mu d,$$

while the rate of deformation tensor is related to the velocity vector by

$$d = \frac{1}{2} \left( \nabla v + (\nabla v)^T \right),$$

where $\sigma \equiv (\sigma_{ij})$ and $d = (d_{ij})$ denote respectively the second order stress tensor and the rate of deformation tensor, $I$ is an identity matrix. As $v = (0, 0, u(r, t))$, we have

$$d = \frac{1}{2} \begin{pmatrix} 0 & 0 & \partial u/\partial r \\ 0 & 0 & 0 \\ \partial u/\partial r & 0 & 0 \end{pmatrix}.$$
From the above formula and using (3.19), we obtain \( d_{rr} = d_{\theta\theta} = d_{zz} = d_{r\theta} = d_{\theta z} = 0 \) and
\[
d_{rz} = -\frac{a_0 R}{8\pi} \left[ -\frac{2r}{R} + \frac{1}{ln\left(\frac{R}{r}\right) + \frac{a_1}{a^2}} \right]
+ \text{Re} \sum_{n=1}^{\infty} \frac{c_n e^{i\omega t}}{2\mu \omega} \left\{ \frac{N_1(a,R,\beta_n l_1, l_2)}{D(a,R,\beta_n l_1, l_2)} \beta_n J_1(\beta_n r) + \frac{N_2(a,R,\beta_n l_1, l_2)}{D(a,R,\beta_n l_1, l_2)} \beta_n Y_1(\beta_n r) \right\}.
\]
(3.31)

Hence from the constitutive equations (3.28), we obtain
\[
\sigma_{rr} = \sigma_{\theta\theta} = \sigma_{zz} = -p = q(t)z + p_0(t), \quad \sigma_{r\theta} = \sigma_{\theta z} = 0
\]
\[
\sigma_{rz} = -\frac{a_0 R}{4} \left[ -\frac{2r}{R} + \frac{1}{ln\left(\frac{R}{r}\right) + \frac{a_1}{a^2}} \right]
+ \text{Re} \sum_{n=1}^{\infty} \frac{c_n e^{i\omega t}}{\rho \omega} \left\{ \frac{N_1(a,R,\beta_n l_1, l_2)}{D(a,R,\beta_n l_1, l_2)} \beta_n J_1(\beta_n r) + \frac{N_2(a,R,\beta_n l_1, l_2)}{D(a,R,\beta_n l_1, l_2)} \beta_n Y_1(\beta_n r) \right\},
\]
(3.32)
where \( q(t) \) is as given in (3.2) while \( p_0(t) \) is arbitrary and can be chosen to meet certain pressure condition.

### 3.5 Influence of Boundary Slip

With the exact solutions obtained in the previous sections, in this section, we discuss the influences of the slip length on velocity, flow rate and stresses in the fluid. As the solution for a general pressure field given by (3.2) is the superposition of the solution due to the constant pressure gradient and the solutions due to the sine and cosine wave form pressure gradients, without loss of generality, we consider here two different cases of driving pressure fields in this discussion. The first case is for a pressure field with a constant pressure gradient, while the second one is for a pressure field with a sine wave form pressure gradient. For convenience in the discussion, we introduce the following dimensionless variables
\[
r^* = \frac{r}{R}, \quad k = \frac{a}{R}, \quad l = \frac{l_2}{R}, \quad \lambda = \frac{l_1}{l_2}, \quad t^* = \frac{\omega t}{2\pi}, \quad \beta^* = \beta R.
\]
(3.33)

**Case 1:** \( \frac{dp}{dz} = a_0. \)

For this case, \( c_n = 0 \) for all \( n \geq 1. \) Then from (3.19), (3.22)-(3.23) and (3.32),
we obtain the following normalized velocity, normalized flow rate and shear stress

\[ u^* = -\frac{4\mu}{a_0 R^2}u = \left[1 - r^{*2} + 2l + \frac{1 - k^2 + 2(1 + k\lambda)l}{ln \frac{1}{k} + (1 + \frac{2}{k})l} (ln r^* - l) \right], \quad (3.34) \]

\[ Q^* = -\frac{8\mu}{a_0\pi R^4}Q = (1 - k^2)(1 - k^2 + 4l) - \frac{1 - k^2 + 2(1 + k\lambda)l}{ln \frac{1}{k} + (1 + \frac{2}{k})l} \left[(1 - k^2)(1 + 2l) + 2k^2\ln(k)\right], \quad (3.35) \]

\[ \sigma_{rz}^* = -\frac{4}{a_0 R} \sigma_{rz} = \left[-2r^* + \frac{1 - k^2 + 2(1 + k\lambda)l}{ln \frac{1}{k} + (1 + \frac{2}{k})l} \frac{1}{r^*}\right]. \quad (3.36) \]

It should be addressed here that for \( l = 0 \), the solution (34) reduces to the solution for the no-slip case [40]. To study the influence of \( l \) on the velocity, we examine the derivative of \( u^* \) with respect to \( l \). From (3.34), we have

\[ \frac{du^*}{dl} = [a(k, \lambda, r^*) + b(k, \lambda)l + c(k, l)]^2 \left[ln \frac{1}{k} + \left(1 + \frac{\lambda}{k}\right)l\right]^{-2}, \quad (3.37) \]

where

\[ a(k, \lambda, r^*) = (2\ln(k) + 1 - k^2)(\ln(k) - \ln(r^*)) - \lambda k\ln(r^*)(2\ln(k) + \frac{1}{k^2} - 1), \]

\[ b(k, \lambda) = 4\lambda\ln(k)(k - \frac{1}{k}), \quad c(k, \lambda) = 2\lambda(\frac{1}{k} - k)(1 + \frac{1}{k}). \]

As \( \lambda > 0, 0 < k < 1, \) and \( k \leq r^* \leq 1, \) we can easily prove that \( a > 0, b > 0 \) and \( c > 0, \) and hence \( \frac{du^*}{dl} > 0 \) for \( l \geq 0, \) which means that the velocity is a monotonically increasing function of \( l. \) Similarly, we can also prove that the flow rate also increases monotonically as \( l \) increases from zero. Also, for \( l \gg \max \left\{1, ln \frac{1}{k}\right\}, \) from (35), we have

\[ \frac{\partial Q^*}{\partial l} \approx 4 (1 - k^2) \left(1 - \frac{1 + k\lambda}{1 + \lambda/k}\right) > 0, \]

which indicates that \( Q^* \) increases almost linearly with increasing \( l \) when \( l \) is sufficient large.

To further demonstrate the characteristics of the variation of the flow rate with \( l \) and \( k, \) we show the solution (3.35) graphically in Figure 2 for the case where both
Figure 3.2: Variation of the flow rate with $l$ and $k$ obtained from solution (3.35) with $\lambda = 1$: (a) 3D graph for $Q^*(k, l)$; (b) contour plot of $Q^*(k, l)$ on $(k, l)$ plane.

the inner and outer surfaces have the same smoothness, i.e. $\lambda = 1$. The result shows that there exists different $(l, k)$ parameter designs for obtaining a given fixed flow rate, which opens a way for the optimal design of the annual. The result also shows that the influence of $l$ on the flow rate is more significant for lower $k$ values. It is also interesting to note that the velocity field in the annual is not a simple superposition of the no-slip solution and a rigid body translation, as is in the circular microtubes [90].

Case 2: $\frac{dp}{dz} = b_1 \sin(\omega t)$.

For this case, $a_0 = 0$, $c_1 = -b_1 i$, $c_n = 0$ for all $n \geq 2$. As $\beta^2 = -\frac{\rho \omega}{\mu} i = \frac{\rho \omega}{\mu} e^{-\pi i/2}$, we have

$$\beta = \sqrt{\frac{\rho \omega}{2\mu}} (1 - i) = \frac{\bar{\beta}}{R} (1 - i), \quad \frac{1}{\beta} = \frac{R}{2\bar{\beta}} (1 + i), \quad \beta^* = \bar{\beta}(1 - i), \quad (3.38)$$

where $\bar{\beta} = R \sqrt{\frac{\rho \omega}{2\mu}}$ is a dimensionless parameter. Then, using the dimensionless variables in (3.33), we have from (3.19), (3.27) and (3.32) that

$$u^* = -\frac{\bar{\beta}}{b_1} u,$$

$$= \Re \left\{ \left[ \frac{N_1}{D} J_0(\beta^* r^*) + \frac{N_2}{D} Y_0(\beta^* r^*) - 1 \right] \frac{1}{2} e^{2\pi t^* i} \right\}, \quad (3.39)$$
\[ Q^* = -\frac{\rho}{2\pi b_1 \mu} Q_n, \]
\[ = \Re \left\{ \left[ \frac{N_1}{\beta J} J_1(\beta^*) - k J_1(\beta^*) k \right] + \frac{N_2}{\beta D} [Y_1(\beta^*) - k Y_1(\beta^*)] \right\} \]
\[ - \frac{1}{2} (1 - k^2) \left[ \frac{1}{\omega} e^{2\pi \omega t} \right], \]
where
\[ \sigma_{rz}^* = \frac{\mu R}{\mu b_1} \sigma_{rz}, \]
\[ = \Re \left\{ \left[ \frac{N_1}{\beta J} \beta^* J_1(\beta^*) + \frac{N_2}{\beta D} \beta^* Y_1(\beta^*) \right] \frac{1}{\omega} e^{2\pi \omega t} \right\}, \] (3.41)

then equation (40) can be written as
\[ Q^* = Q_m^*(k, l) \sin(2\pi t^* + \phi(k, l)), \]

where \(Q_m^*(k, l)\) and \(\phi(k, l)\) are respectively the amplitude and phase angle of the normalized transient flow rate defined by
\[ Q_m^*(k, l) = \left( a - \frac{1}{2\omega} (1 - k^2) \right)^2 + b^2 \right)^{1/2}, \]
\[ \phi(k, l) = \arctan \left( \frac{a - (1-k^2) - a}{b} \right). \] (3.44)

In the following, we first study the influence of the angular frequency \(\omega\) on the amplitude of the transient flow rate \(Q_m^*(k, l)\), and compare the transient solutions with the quasi steady flow rate which is obtained by neglecting the time derivative term in equation (1) to yield
\[ u^* = -\frac{\rho}{b_1} u = \frac{\rho R^2}{4\mu} \left[ 1 - r^*2 + 2l + \frac{1 - k^2 + 2(1 + k\lambda)}{\ln \frac{1}{k} + (1 + \frac{1}{k})} (\ln r^* - l) \right] \sin(2\pi t^*), \] (3.45)
and

\[ Q_s^* = -\frac{\rho}{2\pi b R^2} Q, \]

\[ = \frac{\rho R^2}{16\mu} \left\{ (1 - k^2)(1 - k^2 + 4l) - \frac{1-k^2+2(1+k\lambda)l}{ln(1+\frac{k}{\lambda}l)} \right\} \times \sin(2\pi t^*). \]

(3.46)

Figure 3.3: Variation of the amplitude of the flow rate \( Q_m^* \) with \( l \) and \( k \) obtained for different frequencies \( \omega = \alpha \mu / \rho R^2 \) with (a) \( \alpha = 0.005 \), (b) \( \alpha = 0.00025 \), and (c) \( \alpha = 0.00005 \).

Figure 3.3 shows the variations of the amplitude of the flow rate with \( l \) and
Figure 3.4: Influence of $l$ on the quasi steady-state solution $Q^*_s$ and the transient solution $Q^*_m$ under different frequencies $\omega = \alpha \mu / \rho R^2$ with five different $\alpha$ values: $\alpha = 5.0 \times 10^{-3}$ (dash-box line), $\alpha = 1.0 \times 10^{-3}$ (dash-circle line), $\alpha = 5.0 \times 10^{-4}$ (dash-diamond line), $\alpha = 2.5 \times 10^{-4}$ (dash-cross line), $\alpha = 5.0 \times 10^{-5}$ (dash line).

$k$ for different angular frequencies. Figure 3.4 shows the influence of $l$ on $Q^*_m$ for different angular frequency $\omega$. Obviously, as the angular frequency decreases, the amplitude of the transient flow rate increases and the transient solution converges toward the quasi steady-state solution. It is also noted that the dependence of the amplitude of the transient flow rate on $l$ is very different for different frequencies. At high frequency, the amplitude of the flow rate increases initially as $l$ increases but tends to a constant value once $l$ becomes sufficiently large. On the other hand, at low frequency the flow rate continues to increase with $l$ and depends on $l$ almost linearly for large $l$ values.

Next, as an illustration, we investigate in more detail the influence of $k$ and $l$ on the flow rate for the frequency, $\omega = 5.0 \times 10^{-4} \mu / \rho R^2$, corresponding to a state that is not so close to the quasi steady state as shown by the dash-diamond line in Figure 3.4. We consider here the case where both the inner and outer surfaces have the same smoothness, i.e $\lambda = 1$. For this case, (3.44) give the amplitude of the normalized flow rate as a function of the variables $l$ and $k$ which is shown graphically in Figure 3.5. From the results, various findings can be obtained.
Figure 3.5: Variation of the amplitude of the flow rate with $l$ and $k$ obtained from solution (44) with $\lambda = 1$, $R = 1.0 \times 10^{-5}$, $\rho = 1060$, $\mu = 0.001$, $\omega = 5.0 \times 10^{-4} \mu/\rho R^2$: (a) 3D graph for $Q^*_m$; (b) contour plot of $Q^*_m$ on $(k, l)$ plane.

(i) Unlike for the case of constant pressure gradient, the flow rate in this case no longer increases linearly with $l$ for large $l$ values. For each fixed $k$ value, as $l$ increases, the flow rate increases first and then tends to a constant once $l$ becomes large. The critical $l$ value at which the amplitude of the flow rate tends to a constant value decreases with the increase of the $k$ value.

(ii) The amplitude of the flow rate decreases as $k$ increases, as shown in Figure 3.6. This is because the increase of $k$ not only reduces the cross-section area for the fluid flow, but also leads to lower slip velocity on the solid surface as shown in Figure 3.7.

(iii) As for the constant case, one could have different $(k, l)$ designs to achieve a given flow rate and Figure 3.5(b) provides a tool for the design.

(iv) The influence of $l$ on the flow rate becomes less and less significant as $k$ increases.
Figure 3.6: Variation of the amplitude of the flow rate with $l$ for $\omega = 5.0 \times 10^{-4} \mu/\rho R^2$ and various different $k$ values in case 2: $k = 0.1$ (solid line), $k = 0.3$ (dash line), $k = 0.5$ (dash-cross line), $k = 0.7$ (dash-box line), and $k = 0.9$ (solid-circle line).

Figure 3.7: Variations of the amplitude of the slip velocity $u_m^*$ on the outer surface $r^* = 1$ with $l$ for $\omega = 5.0 \times 10^{-4} \mu/\rho R^2$ and various different $k$ values in case 2: $k = 0.1$ (solid line), $k = 0.3$ (dash line), $k = 0.5$ (dash-cross line), $k = 0.7$ (dash-box line), and $k = 0.9$ (solid-circle line).
3.6 Concluding Remarks

In this section, we derive the exact solutions for the pressure gradient-driven transient flow of an incompressible Newtonian liquid through a circular annual with a Navier slip boundary. Based on the analytical expressions of the solutions, we analyze the influence of the slip parameter $l$ and the geometry of the cross-section on the flow rate of fluid through the annual. The study shows that

(i) The influence of boundary slip on the flow through the annual is different for different types of pressure gradients. For flows driven by a constant pressure gradient, the flow rate always increases with the slip parameter $l$ and achieves a linear increase rate for large $l$ value; while, for the flows driven by the wave form pressure gradient with high frequency, the flow rate initially increases significantly as $l$ increases from zero but tends to a constant when $l$ becomes sufficiently large.

(ii) To achieve a fixed value of flow rate, one could have different $(k, l)$ designs. The exact solutions obtained in this section, together with the contour plots of the solutions (Figure 3.2(b) and Figure 3.5(b)), provide a tool for engineers and scientists to determine the proper $(k, l)$ values.
Chapter 4

Simulation of Particle-fluid Flow

4.1 General Overview

In this chapter, we present a mathematical model to describe the particle-fluid flow in micro-channels. The fluid is assumed to be Newtonian and the motion of particles is assumed to follow Newton’s laws of motion. The finite element method, based on the Arbitrary Lagrangian-Eulerian approach, is used to solve the problem. The stress and velocity fields in fluids and the particle movement are presented and discussed.

The rest of the chapter is organized as follows. The complete set of equations for the fluid-particle flow, based on the Newtonian model, is presented in section 4.2. In section 4.3, an Arbitrary Lagrangian-Eulerian formulation for the mesh movement is presented. In section 4.4, a numerical scheme for solving the problem is developed based on the Bobnov-Galerkin finite element method and the Arbitrary Lagrangian-Eulerian method. In section 4.5, the solution procedure for the problem is given. In section 4.6, a numerical investigation on the stress and velocity fields in fluids is presented followed by the numerical investigation on the movement of particles in section 4.7.

4.2 Formulation of the Problem

In this section, the mathematical model for the fluid flow and particle motion in fluids is described. The governing equations for the fluid flow include the Navier-Stokes equations and the continuity equation. The governing equations for particle
motion are Newton’s second law of motion. The particle-particle interaction and particle-wall interaction are taken into account.

To study the motion of solid particles immersed in a fluid, we assume that the fluid-solid particle system occupies a bounded domain \( \tilde{\Omega} \) in \( \mathbb{R}^3 \). At a typical instant of time \( t \), \( Q \) particles occupy \( Q \) closed connected subsets \( \sum_{q=1}^{Q} \Omega_q \subset \mathbb{R}^3 \) which are surrounded by a viscous homogeneous fluid filling the domain \( \tilde{\Omega} - \sum_{q=1}^{Q} \Omega_q \) called the flow-channel area.

In this study we use two coordinate systems: a reference system, \( \Omega \), where the model is drawn and the particle movement is solved, and a moving mesh system, \( \Omega_{def} \), corresponding to the deformed mesh of the flow channel, where we simulate the fluid flow. The time evolution of the domain \( \Omega_{def} \) is determined by means of an Arbitrary Lagrangian-Eulerian (ALE) mapping \( x : \Omega \times \mathbb{R}^+ \mapsto \Omega_{def} \) which maps any point \((X, t)\) in \( \Omega \) to its image \( x(X, t) \) in \( \Omega_{def} \).

Now we will formulate the mathematical model to describe the fluid flow and particle motion in fluids. Firstly, consider the model for fluid flow. Fluid is assumed to be an incompressible Newtonian fluid. The flow of fluid is described by the continuity equation and the Navier-Stokes equations, in terms of total derivatives,

\[
\nabla \cdot \mathbf{u} = 0, \\
\rho_f \frac{D\mathbf{u}}{Dt} - \nabla \cdot \mathbf{\sigma} = \mathbf{F},
\]

for \( \mathbf{x} = (x, y, z) \in \Omega_{def}(t) \) where \( \rho_f \) denotes the blood density, \( \mathbf{u} = [u_i]_{i=1,2,3} = [u, v, w]^T \) represents the 3D velocity vector, and \( \mathbf{F} \) is the volume force acting on the fluid. For this model, we neglect the effect of gravitational force and thus \( \mathbf{F} = 0 \). The quantity \( \mathbf{\sigma} \) in equation (4.2) is the viscous stress tensor given by

\[
\mathbf{\sigma} = -p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T),
\]

where \( \mu \) is the fluid viscosity, \( p \) is the fluid pressure, and \( \delta_{ij} \) is the Kronecker symbol. The indicial expression of the equations (4.1)-(4.3) are

\[
\frac{\partial u_j}{\partial x_j} = 0,
\]
\[
\rho f \frac{Du_i}{Dt} - \frac{\partial \sigma_{ij}}{\partial x_j} = 0, \quad (4.5)
\]

\[
\sigma_{ij} = -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right). \quad (4.6)
\]

Two types of boundary conditions on the wall will be investigated in the work, including the no-slip condition and the slip condition. Both types of boundary conditions can be written as

\[
u = -l \frac{\partial u}{\partial y} \text{ on } \Gamma_w, \quad (4.7)
\]

where \(l\) is the slip parameter. It should be addressed here that \(l = 0\) corresponds to the no-slip case.

The flow in microscale is known to be laminar profile. Thus the inlet boundary condition is assumed to be laminar, which is given by,

\[
-L_{\text{enter}} \nabla_t \cdot (-p \mathbf{I} + \mu (\nabla_t \mathbf{u} + (\nabla_t \mathbf{u})^T) = -p_{\text{enter}} \mathbf{n} \text{ on } \Gamma_{\text{in}},
\]

\[
\nabla_t \cdot \mathbf{u} = 0 \text{ on } \Gamma_{\text{in}}, \quad (4.8)
\]

where \(L_{\text{enter}}\) is the entrance length of the fictitious channel inlet, \(p_{\text{enter}}\) is the entrance pressure at the channel inlet, \(\mathbf{n}\) is the outward normal vector. The pressure \(p_{\text{enter}}\) on the inlet is determined by solving (4.8) such that the computed average velocity of the laminar flow equals to a given mean velocity \(u_{\text{mean}}\), and \(\nabla_t\) denotes that this equation is solved for tangential components on the boundary.

On the outflow boundary \(\Gamma_{\text{out}}\), the stress-free condition is used:

\[
\sigma \cdot \mathbf{n} = 0 \text{ on } \Gamma_{\text{out}}. \quad (4.9)
\]

On the boundary of particles, the velocity of blood is assumed to be the same as that of particles, which is

\[
u = \mathbf{V}_q \text{ on } \Gamma_q, \quad (4.10)
\]

where \(\mathbf{V}_q\) will be determined later.

Therefore the boundary conditions for the velocity field of fluids are of the
Dirichlet type and Neumann type,

\[-L_{\text{enter}} \nabla_t (-pI + \mu(\nabla_t u + (\nabla_t u)^T) = -p_{\text{enter}}n \quad \text{on } \Gamma_{\text{in}},
\]

\[\nabla_t \cdot u = 0 \quad \text{on } \Gamma_{\text{in}},\]

\[u_t = -l \frac{\partial u_t}{\partial y} \quad \text{on } \Gamma_{\text{w}},\]

\[\mu(u_{i,j} + (u_{j,i})) \cdot n = 0 \quad \text{on } \Gamma_{\text{out}},\]

\[u_i = V_q \quad \text{on } \Gamma_{\text{q}}.\]  \hfill (4.11)

Now consider the mathematical model for particle flow. To study the motion of particles in the fluid flow channel, we assume that the gravitational force can be neglected and the particle motion is governed by Newton’s second law:

\[m_q \frac{\partial V_q}{\partial t} = F_v + F_q, \quad q = 1, 2, 3..., Q\]  \hfill (4.12)

The position \(X_q\) of the center of the \(q\)th particle can be determined by the equation:

\[\frac{dX_q}{dt} = V_q, \quad q = 1, 2, 3..., Q\]  \hfill (4.13)

In equation (4.12), \(V_q\) and \(m_q\) denote, respectively, the velocity vector and the mass of the \(q\)th particle. Next, we will determine the two applied loads, drag force \(F_v\) and collision force \(F_q\). For the drag force \(F_v\) acting on particle \(\Omega_q\), it is assumed that all the boundaries of particles experience drag force \(F_v\) from the fluid given by

\[F_v = \int_{\partial \Omega_q} (-n_f \cdot \sigma) dS = -\int_{\partial \Omega_q} n_f \cdot (\nabla u + \eta(\nabla u + (\nabla u)^T)) dS,\]  \hfill (4.14)

where \(n_f\) is the normal vector of the fluid domain. It is seen that the drag force \(F_v\) consists of the pressure and the viscous drag of the fluid. To prevent collision among the particles and the collision between the particles and vessel walls, the particle-particle interaction force \(F_{q,p}\) and the particle-wall interaction force \(F_{q,w}\) are applied when the distance between two particles, or between a particle and a wall, is within the order of the element size \([70]\). Thus

\[F_q = \sum_{p=1, p \neq q}^Q F_{q,p} + \sum_{w=1}^2 F_{q,w},\]  \hfill (4.15)
in which

\[
F_{q,p} = \begin{cases} 
0, & \text{for } d_{q,p} > R_q + R_p + \alpha \\
\frac{1}{\varepsilon_q} (X_q - X_p)(R_q + R_p + \alpha - d_{q,p})^2, & \text{for } d_{q,p} \leq R_q + R_p + \alpha 
\end{cases} \tag{4.16}
\]

and

\[
F_{q,w} = \begin{cases} 
0, & \text{for } d_{q,w} > 2R_q + \alpha \\
\frac{1}{\varepsilon_w} (X_q - X_w)(2R_q + \alpha - d_{q,w})^2, & \text{for } d_{q,w} \leq 2R_q + \alpha 
\end{cases} \tag{4.17}
\]

where \(d_{q,p}\) denotes the distance between the centers of the \(q\)th and \(p\)th particles, \(d_{q,w}\) denotes the distance between the centers of the \(q\)th particle and the imaginary particle on the other side of the wall, \(X_q\) and \(R_q\) are respectively the center and radius of the \(q\)th particle, \(\alpha\) is the force range, and \(\varepsilon_q\) and \(\varepsilon_w\) are small positive stiffness parameters.

### 4.3 Arbitrary Lagrangian-Eulerian (ALE) Mesh Movement

In order to construct a mesh-based numerical model involving the motion of particles in fluids, we will introduce the Arbitrary Lagrangian-Eulerian method which will involve the mesh movement of the numerical model.

We follow the ALE description given by [96]. The method is restated as follow. Let \(\phi\) be a scalar variable of \(x\) in \(\Omega_{de,f}(t)\) and \(t\). The variable \(\phi\) is transported from a position \(P\) at a time \(t\) to a new position \(P_f\) at \(t + \Delta t\) with a velocity of \(u\). The mesh point is moved to a new position \(P_r\) at \(t + \Delta t\) with a mesh velocity of \(\Psi\). Using Taylor series expansion in time, we calculate \(\phi\) for position \(P_f\) at time \(t + \Delta t\) by

\[
\phi_{P_f}^{n+1} = \phi_{P_f}^n + \Delta t \frac{\partial \phi_{P_f}^n}{\partial t} + \ldots \tag{4.18}
\]

Using Taylor series expansion in space, we express \(\phi\) for position \(P_f\) at time \(t\) by

\[
\phi_{P_f}^n = \phi_P^n + u_i \Delta t \frac{\partial \phi_P^n}{\partial x_i} + \ldots, \tag{4.19}
\]

where \(x_i = x, y, z\) for \(i = 1, 2, 3\), respectively.
Substituting (4.21) into (4.18), we have
\[ \phi_{P_n} = \phi_{P}^{n+1} = \phi_{P}^{n} + u_i \Delta t \frac{\partial \phi_{P}^{n}}{\partial x_i} + \Delta t \frac{\partial}{\partial t} \left( \phi_{P}^{n} + u_i \Delta t \frac{\partial \phi_{P}^{n}}{\partial x_i} \right) + \ldots, \] 
(4.20)
Similarly, we express \( \phi \) at position \( P_r \) at time \( t \), using Taylor series expansion in time, as
\[ \phi_{P_r} = \phi_{P_r}^{n} = \phi_{P_r}^{n} + \Psi_i \Delta t \frac{\partial \phi_{P_r}^{n}}{\partial x_i} + \ldots, \] 
(4.21)
where \( \Psi_i = \Psi_x, \Psi_y, \Psi_z \) for \( i = 1, 2, 3 \), respectively.

Neglecting second and higher order terms, the relative value of \( \phi \) between the actual particle and the mesh motion is then given by
\[ \Delta \phi = \phi^{n+1} - \phi^n = \Delta t \frac{\partial \phi_{P}^{n}}{\partial t} + (u_i - \Psi_i) \Delta t \frac{\partial \phi_{P}^{n}}{\partial x_i}. \] 
(4.22)
As \( \Delta t \to 0 \) and dropping super and subscripts, (4.22) can be rewritten as
\[ \frac{D \phi}{D t} = \frac{\partial \phi}{\partial t} + (u_i - \Psi_i) c, \] 
(4.23)
or
\[ \frac{D \phi}{D t} = \frac{\partial \phi}{\partial t} + (u - \Psi) \cdot \nabla \phi. \] 
(4.24)
These total derivatives in the ALE form will be used to describe the motion of material in the ALE frame.

In the ALE coordinate system, the continuity equation (4.1) and the Navier-Stokes equations (4.2) can be, using the total derivatives in the ALE form (4.24), written as,
\[ \frac{\partial u_i}{\partial x_i} = 0, \] 
(4.25)
\[ \rho f \frac{\partial u_i}{\partial t} + \rho f (u_j - \Psi_j) \frac{\partial u_i}{\partial x_j} - \frac{\partial \sigma_{ij}}{\partial x_j} = 0. \] 
(4.26)

Due to the movement of the coordinate system, the mesh velocity \( \Psi = (\Psi_x, \Psi_y, \Psi_z) \) is introduced in the deformed domain \( \Omega_{def} \). To guarantee a smoothly varying distribution of the nodes, we assume that the nodes on \( \partial \Omega_q \) move with the particle (no slip) and that each component of the mesh velocity in the fluid channel is governed by a Laplace equation:
\[ \nabla^2 \Psi = 0, \quad \forall x \in \Omega_{def}. \] 
(4.27)
The above equation is to smooth gradient of the mesh velocity over the domain so as to reduce mesh distortion. Once the mesh velocity components are determined, we can determine the smoothed deformed mesh for the flow channel at each time instant by updating the coordinates of the nodes according to the following formulae

\[
x = X + \int_{0}^{t} \Psi_x \, dt, \\
y = Y + \int_{0}^{t} \Psi_y \, dt, \\
z = Z + \int_{0}^{t} \Psi_z \, dt.
\] (4.28)

Another condition that needs to be specified is that the fluid, particle and mesh all move with the same velocity on the particle boundaries, i.e.,

\[
\Psi = u = V_q \text{ on } \partial \Omega_q.
\] (4.29)

### 4.4 Finite Element Formulation

In order to study the flow of particles, we implement the finite element approach based on the Arbitrary Lagrangian-Eulerian method.

Substituting the stress tensor term from equation (4.6) into equation (4.26), we obtain the following Navier–Stokes equations

\[
\rho_f \left( \frac{\partial u_i}{\partial t} + (u_j - \Psi_j)u_{i,j} \right) - \left( \mu (u_{i,j} + u_{j,i}) \right)_{,j} + p_{,i} = 0.
\] (4.30)

Note that a new variable \( \Psi_j \) of mesh velocity is added to the Navier–Stokes equations.

It is seen that these Navier–Stokes equations together with the continuity equation (4.25) constitute a closed system of seven partial differential equations in terms of seven coordinate and time–dependent unknown functions \( u_1, u_2, u_3, p, \Psi_1, \Psi_2, \text{ and } \Psi_3 \).

Now we have a complete set of equations to describe the fluid flow and particle flow in fluid. The governing equations consist of the continuity equation (4.25), the Navier-Stokes equations (4.30), and the Laplace equation for mesh smoothing (4.27). For convenience in formulation, we rewrite equations (4.27) in the indicial notation as follows,

\[
\Psi_{i,jj} = 0.
\] (4.31)
The field equations (4.25), (4.30), and (4.31) are defined in \( \Omega_{def}(t) \) with the boundary conditions (4.11) and (4.29). These equations and boundary conditions constitute a closed system of seven partial differential equations in terms of seven unknown functions \( u_i, p, \Psi_i \). The system, supplemented by the initial condition and the boundary conditions (4.11) and (4.29), can be solved numerically to yield the velocity field and the pressure distribution and consequently the shear stresses on the wall. Thus, the boundary value problem for the fluid-particle flow problem is as follows:

**BVP:** Find \( u_i, p, \Psi_i \) such that the field equations (4.25), (4.30), and (4.31) are satisfied in \( \Omega_{def}(t) \) and all boundary conditions (4.11) and (4.29) are satisfied.

To derive a variational statement for the BVP, we consider the following associated variational boundary value problem.

**VBVP:** Find \( u_i, p, \Psi_i \in H^1(\Omega_{def}) \) in the deformed mesh system at each time instant such that all the Dirichlet boundary conditions (4.11) and (4.29) are satisfied and for all \( \hat{u}_i \in H^1(\Omega_{def}) \equiv \{ \hat{u}_i \in H^1(\Omega_{def}) | \hat{u}_i = 0 \text{ on } \partial \Omega_{def}\} \), \( \hat{p} \in H^1(\Omega_{def}) \equiv \{ \hat{p} \in H^1(\Omega_{def}) | \hat{p} = 0 \text{ on } \partial \Omega_{def}\} \), \( \hat{\Psi}_i \in H^1(\Omega_{def}) \equiv \{ \hat{\Psi}_i \in H^1(\Omega_{def}) | \hat{\Psi}_i = 0 \text{ on } \partial \Omega_{def}\} \), and

\[
(u_{i,i}, \hat{p}) = 0, \tag{4.32}
\]

\[
\left( \rho_f \frac{\partial u_i}{\partial t}, \hat{u}_i \right) + \left( \rho_f (u_j - \Psi_j) u_{i,j}, \hat{u}_i \right) - \left( \mu (u_{i,j} + u_{j,i})_{,j}, \hat{u}_i \right) + \left( p_{,i}, \hat{u}_i \right) = 0, \tag{4.33}
\]

and

\[
(\Psi_{i,jj}, \hat{\Psi}_i) = 0, \tag{4.34}
\]

where \( \partial \Omega_{defu}, \partial \Omega_{defp}, \) and \( \partial \Omega_{def\Psi} \) are the parts of boundary where the velocity, the pressure, and the mesh velocity are specified. \( H^1(\Omega_{def}) \) is the Sobolev space \( W^{1,2}(\Omega_{def}) \) with norm \( \| \cdot \|_{1,2,\Omega_{def}} \) and the inner product \( (\cdot, \cdot) \) is defined by

\[
(a, b) = \int_{\Omega_{def}} (ab) d\Omega. \tag{4.35}
\]
Using the continuity equation, the integration by parts technique, the divergence theorem, and the boundary condition, we can eliminate the second order derivatives in (4.33) and (4.34) to obtain

\[
\left( \rho_f \frac{\partial u_i}{\partial t} , \hat{u}_i \right) + \left( \rho_f (u_j - \Psi_j) u_{i,j} , \hat{u}_i \right) + \mu \left( u_{i,j} \hat{u}_{i,j} \right) + \left( p_x, \hat{u}_i \right) - \mu \left( u_{i,j} n_j, \hat{u}_i \right)_B = 0,
\]

and

\[
(\Psi_{i,j}, \dot{\Psi}_{i,j}) = 0,
\]

where \((a, b)_B\) denotes integration of \(ab\) over the boundary of \(\Omega\) except for the part with Dirichlet boundary condition.

Since the computations are conducted in the reference coordinates, \(\Omega\), we need to transform equations (4.32), (4.36), and (4.37) in the deformed coordinates to those equations in the reference coordinates. Through this, the inner product expression in (4.35) is re-defined as

\[
(a, b) = \int_{\Omega_{def}} (ab) \, d\Omega = \int_{\Omega} (ab) \, |J| \, d\Omega,
\]

where the so-called Jacobian matrix \(J\) and the derivatives of the unknown functions \(u_i, p\) and \(\Psi_i\) in equations (4.32), (4.36), and (4.37) are defined as follows.

In the flow-channel area, the two coordinate systems, \((X, Y, Z) \in \Omega\) and \((x, y, z) \in \Omega_{def}\), are connected through a transformation \(T\). At the initial state at \(t = 0\), the two mesh systems are assumed to coincide. The transformation \(T\) maps the point initially located at \((X, Y, Z)\) to the point \((x, y, z)\) at time \(t\):

\[
T : \begin{align*}
x &= x(X, Y, Z, t) \\
y &= y(X, Y, Z, t) \\
z &= z(X, Y, Z, t).
\end{align*}
\]

Suppose that the functions \(x, y\) and \(z\) are continuous differentiable with respect to \(X, Y, Z\). Then the infinitesimals \(dX, dY, dZ\) transform into \(dx, dy, dz\) according to

\[
\begin{align*}
dx &= x_X dX + x_Y dY + x_Z dZ, \\
dy &= y_X dX + y_Y dY + y_Z dZ, \\
dz &= z_X dX + z_Y dY + z_Z dZ,
\end{align*}
\]

(4.39)
where (·),X denotes differentiation with respect to X. System (4.39) can be written in matrix form as

\[
\begin{bmatrix}
\frac{dx}{dt} \\
\frac{dy}{dt} \\
\frac{dz}{dt}
\end{bmatrix} =
\begin{bmatrix}
x, x & x, y & x, z \\
y, x & y, y & y, z \\
z, x & z, y & z, z
\end{bmatrix}
\begin{bmatrix}
\frac{dX}{dt} \\
\frac{dY}{dt} \\
\frac{dZ}{dt}
\end{bmatrix}.
\]

(4.40)

The 3 × 3 matrix of partial derivatives in (4.40) is called the Jacobian matrix of the transformation. Denote the matrix by J, then

\[
|J| = x, x(y, y z, z - y, z z, y) - x, y(y, x z, z - y, z z, x) + x, z(y, x z, y - y, y z, x).
\]

For |J| ≠ 0, the transformation is invertible and there exists an inverse transformation at time t, i.e.,

\[
T^{-1} : \begin{align*}
X &= X(x, y, z) \\
Y &= Y(x, y, z) \\
Z &= Z(x, y, z).
\end{align*}
\]

As in (4.40), we have

\[
\begin{bmatrix}
\frac{dX}{dt} \\
\frac{dY}{dt} \\
\frac{dZ}{dt}
\end{bmatrix} =
\begin{bmatrix}
X, x & X, y & X, z \\
Y, x & Y, y & Y, z \\
Z, x & Z, y & Z, z
\end{bmatrix}
\begin{bmatrix}
\frac{dx}{dt} \\
\frac{dy}{dt} \\
\frac{dz}{dt}
\end{bmatrix}.
\]

(4.41)

From (4.40), we also have

\[
\begin{bmatrix}
\frac{dX}{dt} \\
\frac{dY}{dt} \\
\frac{dZ}{dt}
\end{bmatrix} = J^{-1}
\begin{bmatrix}
\frac{dx}{dt} \\
\frac{dy}{dt} \\
\frac{dz}{dt}
\end{bmatrix},
\]

(4.42)

where

\[
J^{-1} = \frac{1}{|J|}
\begin{bmatrix}
y, y z, z - y, z z, y & x, z z, y - x, y z, z & x, y y, z - x, y z, y \\
y, z z, x - y, x z, z & x, z z, x - x, z z, x & y, x y, z - y, z x, x \\
y, y z, y - y, y z, z & x, y z, x - x, y z, y & x, x y, y - x, y y, x
\end{bmatrix},
\]

(4.43)
Equating terms in (4.41) and (4.42), we obtain

\[ X_{x} = \frac{1}{\Omega} (y, y z, z - y, z z, y), \]
\[ X_{y} = \frac{1}{\Omega} (x, z z, y - x, y z, z), \]
\[ X_{z} = \frac{1}{\Omega} (x, y y, z - x, z y, y), \]
\[ Y_{x} = \frac{1}{\Omega} (y, z z, x - y, x z, z), \]
\[ Y_{y} = \frac{1}{\Omega} (x, x z, z - x, z z, x), \]
\[ Y_{z} = \frac{1}{\Omega} (y, x x, z - y, z x, x), \]
\[ Z_{x} = \frac{1}{\Omega} (y, x z, y - y, y z, x), \]
\[ Z_{y} = \frac{1}{\Omega} (x, y z, x - x, x z, y), \]
\[ Z_{z} = \frac{1}{\Omega} (x, x y, y - x, y y, x). \]

(4.44)

These relations are crucial in transforming the calculation results from \( \Omega_{def} \) to \( \Omega \).

Now the derivatives of the unknown functions \( \varphi_i (i = x, y, z) \), where \( \varphi_i \) represents \( u_i, p \), and \( \Psi_i \), can be determined by the following expressions:

\[ \varphi_{i,x} = \varphi_{i,x} X_{x} + \varphi_{i,y} Y_{x} + \varphi_{i,z} Z_{x}, \]
\[ \varphi_{i,y} = \varphi_{i,x} X_{y} + \varphi_{i,y} Y_{y} + \varphi_{i,z} Z_{y}, \]
\[ \varphi_{i,z} = \varphi_{i,x} X_{z} + \varphi_{i,y} Y_{z} + \varphi_{i,z} Z_{z}, \]

(4.45)

or

\[ \varphi_{i,j} = \varphi_{i,x} X_{k,j}, \]

(4.46)

and for the test functions:

\[ \hat{\varphi}_{i,x} = \hat{\varphi}_{i,x} X_{x} + \hat{\varphi}_{i,y} Y_{x} + \hat{\varphi}_{i,z} Z_{x}, \]
\[ \hat{\varphi}_{i,y} = \hat{\varphi}_{i,x} X_{y} + \hat{\varphi}_{i,y} Y_{y} + \hat{\varphi}_{i,z} Z_{y}, \]
\[ \hat{\varphi}_{i,z} = \hat{\varphi}_{i,x} X_{z} + \hat{\varphi}_{i,y} Y_{z} + \hat{\varphi}_{i,z} Z_{z}, \]

(4.47)

or

\[ \hat{\varphi}_{i,j} = \hat{\varphi}_{i,x} X_{k,j}. \]

(4.48)

Using (4.46) and (4.48), equations (4.32)-(4.34) become

(4.49)

\[ \left( \rho f \frac{\partial u_i}{\partial t}, \hat{u}_i \right) + \rho_f (u_j - \Psi_j) u_i, X_{k,j}, \hat{u}_i \]  
\[ + \mu \left( u_i, X_{k,j}, \hat{u}_i, X_{k,j} \right) + \rho_s X_{k,i}, \hat{u}_i - \mu \left( u_i, X_{k,j}, \hat{u}_i \right)_B = 0, \]

(4.50)
(\Psi_{i,X_k} X_{k,j}, \hat{\Psi}_{i,X_k} X_{k,j}) = 0. \tag{4.51}

To get the solution of the above VBVP problem, the Bubnov-Galerkin finite element method is used in this work. We choose an \( M \)-dimensional subspace \( H_h \subset H^1(\Omega_{def}) \) for \( u_i, \Psi_i \) and the corresponding test functions, and \( N \)-dimensional subspace for \( H_\beta \subset H^1(\Omega_{def}) \) for \( p \) and the corresponding test functions. Let \( \{\phi_m\}_{m=1}^M \) be the basis functions of \( H_h \), and \( \{\phi_p^n\}_{n=1}^N \) be the basis functions of \( H_\beta \). Then we have

\[
u_i(X, t) \approx (u_i)_h = \sum_{m=1}^M \phi_m(X)(u_i)_m(t). \tag{4.52}
\]

\[
\hat{u}_i(X, t) \approx (\hat{u}_i)_h = \sum_{m=1}^M \phi_m(X)(\hat{u}_i)_m(t). \tag{4.53}
\]

\[
\Psi_i(X, t) \approx (\Psi_i)_h = \sum_{m=1}^M \phi_m(X)(\Psi_i)_m(t). \tag{4.54}
\]

\[
\hat{\Psi}_i(X, t) \approx (\hat{\Psi}_i)_h = \sum_{m=1}^M \phi_m(X)(\hat{\Psi}_i)_m(t). \tag{4.55}
\]

\[
p(X, t) \approx p_h = \sum_{n=1}^N \phi^p_n(X)p_n(t). \tag{4.56}
\]

\[
\hat{p}(X, t) \approx \hat{p}_h = \sum_{n=1}^N \phi^p_n(X)\hat{p}_n(t). \tag{4.57}
\]

Substituting equations (4.52)-(4.57) into equations (4.49)-(4.51) and using the arbitrary of \((\hat{u}_i)_m, (\hat{\Psi}_i)_m, \) and \(\hat{p}_n\), we obtain the following system of equations

\[
\sum_{m=1}^M \int_{\Omega} \phi^p \frac{\partial \phi_m}{\partial X_k} \frac{\partial X_k}{\partial x_i} |J|d\Omega(U_i)_m = 0, \tag{4.58}
\]
\[
\rho_f \sum_{m=1}^{M} \int_{\Omega} \phi_k \phi_m |J| d\Omega (\hat{U}_i)_m \\
+ \rho_f \sum_{m=1}^{M} \int_{\Omega} \phi_k \left( u_j - \Psi_j \right) \frac{\partial \phi_m}{\partial X_p} \frac{\partial X_p}{\partial x_j} |J| d\Omega (U_i)_m \\
+ \sum_{m=1}^{M} \int_{\Omega} \frac{\partial \phi_k}{\partial x_i} \frac{\partial X_i}{\partial x_j} \frac{\partial X_p}{\partial x_j} |J| d\Omega (U_i)_m \\
+ \sum_{n=1}^{N} \int_{\Omega} \mu \frac{\partial \phi_k}{\partial x_i} \frac{\partial X_k}{\partial x_i} |J| d\Omega (P)_n \\
- \sum_{m=1}^{M} \int_{\partial \Omega} \phi_k \frac{\partial \phi_m}{\partial x_i} \frac{\partial X_i}{\partial x_j} |J| dS (U_i)_m = 0, \\
\sum_{m=1}^{M} \int_{\Omega} \frac{\partial \phi_k}{\partial x_i} \frac{\partial X_k}{\partial x_j} \frac{\partial X_i}{\partial x_j} |J| d\Omega (\Psi_i)_m = 0.
\]

(4.59) (4.60)

We then apply boundary conditions (4.11) and (4.29) to equations (4.58)-(4.60). After a mathematical derivation, we obtain a system of ordinary differential equations which can be written in the terms of global vectors of \( u, p, \) and \( \Psi \) with each \( i \)th entry representing the value of the corresponding unknown function at the \( i \)th node of the finite element mesh.

A standard backward Euler scheme is then used to solve the system of ordinary differential equations to determine the velocity and pressure fields at any instant of time.

### 4.5 Solution Procedure

Based on the previous formulation, the solution procedure for solving the fluid flow-particle motion problem is summarized as follows.

Initialization: \( t_0 = 0, n = 0 \) (index for time step),

Generate initial mesh \( x_0 \) based on particle positions, \( X_q^i(0) \).

Initialize \( u(X, t_0) \).

Do \( n = 1, 2, \ldots, N \) (total number of time steps)

1. Select time step \( \Delta t_n : \Delta t_n = t_{n-1} + \Delta t_n \).
2. Update particle positions:
\[ X^i_q(t_n) = X^i_q(t_{n-1}) + V^i_q(t_{n-1}) \Delta t_n. \]

3. Update mesh nodes:
\[ y(t_n) = x(X, t_{n-1}) + \Psi(X, t_{n-1}) \Delta t_n. \]

4. Check mesh quality; if the updated mesh \( y(t_n) \) is not too deformed, then generate a new mesh \( x(t_n) \), and then project the flow field from \( y(t_n) \) onto \( x(t_n) \).

5. Iteratively solve for the flow field \( u(x(t_n), t_n) \), and \( p(x(t_n), t_n) \);
the mesh velocity \( \Psi(x(t_n), t_n) \); and the particle velocities \( V^i_q(t_n) \).

6. Update \( \dot{V}^i_q(t_n) \) from the equation (4.13).

7. Solve the mesh acceleration.

End Do.

4.6 Stress and Velocity Fields in Fluids

We consider here an example to study the flow of a Newtonian fluid passing around a solid cylinder in a microchannel with slip boundary. The cylinder is located at a fixed position in the middle of the channel. The height of the microchannel is 1.2\( \mu \)m, and the length 3\( \mu \)m, while the radius of the cylinder is 0.1\( \mu \)m. Other parameters have the values as follows: fluid density \( \rho_f = 1000 \text{ kg/m}^3 \), viscosity of fluid is \( \mu = 0.001 \text{ kg/(m} \cdot \text{s)} \), slip parameters \( l = 0 \) (no slip), \( 0.25 \times 10^{-7}, 0.5 \times 10^{-7}, 1.0 \times 10^{-7}, \text{ and } 2.0 \times 10^{-7} \).

The finite element method presented in the previous sections is used to simulate the fluid flow through the microchannel at the presence of a particle on the channel. The aim is to investigate the influence of the slip parameter on the flow behaviour. Figure 4.1 shows the computational domain and its finite element mesh consisting of 27439 elements together with the coordinates used.

Figure 4.2 (a–c) shows the profiles of the velocity \( u_x \) on the channel cross-section at three different locations along the channel for five different values of the slip parameter \( l \). The results show that at the upstream and downstream sufficiently
far from the cylinder, the velocity profile is parabolic for \( l = 0 \) (no slip) and tends to uniform as \( l \) increases. It can also be noted that the velocity increases significantly as the slip parameter \( l \) increases.

Figure 4.3 shows the influence of the slip parameter \( l \) on the drag force exerted on the microcylinder surface by the fluid. It is clear from the results that as \( l \) increases, the drag force increases significantly.

Figure 4.4 shows the flow rate of fluid calculated at the outlet end of the microchannel for five different values of slip parameter \( l = 0 \) (or no slip), \( 0.25 \times 10^{-7} \), \( 0.5 \times 10^{-7} \), \( 1.0 \times 10^{-7} \), and \( 2.0 \times 10^{-7} \). The results show that the slip parameter has significantly influences on the flow rate. As \( l \) increases, the flow rate increases.

Figure 4.5 depicts the streamlines and velocity field of flow in the microchannel for five different values of slip parameter \( l = 0 \) (no slip), \( 0.25 \times 10^{-7} \), \( 0.5 \times 10^{-7} \), \( 1.0 \times 10^{-7} \), and \( 2.0 \times 10^{-7} \). The plots show the pattern of fluid flow. In downstream from the cylinder, the velocity profile is similar to that in upstream. As \( l \) increases, the fluid flows faster in the region near the microchannel wall than in the region near the surface of the cylinder.
Figure 4.2: Cross-section profile of the velocity $u_x$ at three different locations along the channel for five different slip parameter values $l = 0$ (or no slip) (solid-box line), $l = 0.25 \times 10^{-7}$ (solid-cross line), $l = 0.5 \times 10^{-7}$ (solid-asterisk line), $l = 1.0 \times 10^{-7}$ (solid-circle line), and $l = 2.0 \times 10^{-7}$ (solid-triangle line): (a) at the location $x = 0.5\,\mu\text{m}$, (b) at the location $x = 1.5\,\mu\text{m}$ (through the center of the microcylinder), and (c) at the location $x = 2.5\,\mu\text{m}$ along the microchannel. The $x$ and $y$ coordinates are shown in Figure 4.1.
Figure 4.3: Influence of the slip parameter $l$ on the drag force exerted on the cylinder by fluid. The circles represent numerical results, while the solid line is the least square fitting of the numerical results.

Figure 4.4: Influence of the slip parameter $l$ on the flow rate of fluid through the microchannel.
Figure 4.5: Streamlines and vector plots of the velocity fields in fluid in the microchannel for five different slip parameter values: (a) $l = 0$ (no slip), (b) $l = 0.25 \times 10^{-7}$, (c) $l = 0.5 \times 10^{-7}$, (d) $l = 1.0 \times 10^{-7}$, and (e) $l = 2.0 \times 10^{-7}$. The scaled bar shows the scale of the magnitude of velocity in m/s.
4.7 Movement of Particles

In the previous section, we study the flow behaviour of a Newtonian fluid passing around a spatially fixed particle in a microchannel and analyse the influence of slip boundary. In this section, we will focus on understanding the particle-fluid interaction and the particle flow driven by the fluid drag force. For this purpose, we consider the flow of a fluid with one, three, five and nine particles immersed in the fluid through a microchannel.

The computational domain is a horizontal channel with height of 6.2 \( \mu \text{m} \) and length of 45 \( \mu \text{m} \). The particles are circular with diameter of 0.5 \( \mu \text{m} \). The fluid is assumed to flow into the channel with speed 0.7 mm/s from the left to the right. The fluid properties are typical of human blood in capillary vessels with the viscosity \( \eta \) of 0.002 Pa \( \cdot \) s and the density \( \rho_f \) of 1040 kg/m\(^3\). All particles are assumed to be solid particles with the material density of 1400 kg/m\(^3\).

The Arbitrary Lagrangian Eulerian approach as presented in section 4.3 is used to handle the dynamics of deforming geometry and the moving boundaries. New mesh coordinates on the channel area are calculated based on the movement of the particles. The Navier-Stokes equations are formulated in the moving coordinate system and are solved by the method presented in section 4.4.

![Image](image.png)

(a) \( t = 0 \text{ sec.} \)

(b) \( t = 1.62 \text{ milli sec.} \)

(c) \( t = 2.43 \text{ milli sec.} \)

Figure 4.6: Velocity profiles at various instants of time for the case with one particle.
Figure 4.7: Velocity profiles at various instants of time for the case with three particles.

(a) $t = 0$ sec.
(b) $t = 0.81$ milli sec.
(c) $t = 1.18$ milli sec.

Figure 4.8: Velocity profiles at various instants of time for the case with five particles.

(a) $t = 0$ sec.
(b) $t = 0.69$ milli sec.
(c) $t = 1.02$ milli sec.

Figure 4.9: Velocity profiles at various instants of time for the case with nine particles.

(a) $t = 0$ sec.
(b) $t = 0.39$ milli sec.
(c) $t = 0.53$ milli sec.
Figures 4.10: Pressure profiles along the flow direction at $t = 0s$.

Figures 4.6–4.9 show the velocity profiles and particle positions at various instants of time for the cases with one, three, five and nine particles immersed in the fluid. Figure 4.10 shows the variation of the fluid pressure along the channel at a typical instants of time for the four cases. Obviously, except for the region near the particles, the velocity basically has the parabolic profile. It is also noted that the inlet pressure required to drive the flow increases significantly with the increase of the particle number. This is reasonable as larger pressure is required to drive the flow of more particles through the channel.

To understand the influence of the particle initial velocity $V_p(t_0)$, we consider three different cases of particle initial velocity, $V_p(t_0) = 0$, $V_p(t_0) = u(t_0)$, and $V_p(t_0) = 2u(t_0)$ where $V_p(t_0)$ is the particle initial velocity and $u(t_0)$ is the fluid velocity on the inlet of the channel.

Figures 4.11–4.13 show the details of the evolution of the particle position, the forces acting on the particles as well as the particle velocity for the three cases of particle initial velocity.

For the case in which the particle initial velocity is zero, as shown in Figure 4.11, the drag force and consequently the total force acting on the particle is positive in the $x$-direction at the initial stage as the fluid velocity is larger than the particle velocity. This results in an acceleration of the particle and hence the particle velocity in the $x$-direction continues to increase and the particle move gradually to
Figure 4.11: Computed results for the fluid-particle flow with $V_p(t_0) = 0$: (a)–(b) the $x$ and $y$ coordinates of the locations of the particle; (c)–(d) the $x$ and $y$ components of the particle velocity along the channel; (e)–(f) the $x$ and $y$ components of the total force acting on the particle; (g) the trajectory the of particle.
Figure 4.12: Computed results for the fluid-particle flow with $V_p(t_0) = u(t_0)$: (a)–(b) the $x$ and $y$ coordinates of the locations of the particle; (c)–(d) the $x$ and $y$ components of the particle velocity along the channel; (e)–(f) the $x$ and $y$ components of the total force acting on the particle; (g) the trajectory the particle.
Figure 4.13: Computed results for the fluid-particle flow with $V_p(t_0) = 2u(t_0)$: (a)–(b) the $x$ and $y$ coordinates of the locations of the particle; (c)–(d) the $x$ and $y$ components of the particle velocity along the channel; (e)–(f) the $x$ and $y$ components of the total force acting on the particle; (g) the trajectory the particle.
the right. When the particle velocity tends to the fluid velocity, the fluid drag force acting on the particle becomes zero and consequently the particle velocity remains constant as shown in figure 4.11 (e).

On the other extreme, when the particle moves much faster than the fluid, the fluid provides a resistance to the movement of the particle resulting in a negative total force acting on the particle in the x-direction. Consequently, the particle velocity decreases. Once the particle velocity decreases toward the fluid velocity, the total force acting on the particle tends to zero and consequently the particle velocity tends to a constant.

4.8 Concluding Remarks

A mathematical model and numerical technique for simulating particle-fluid flow in microchannels has been established. The model is then used to study the influence of boundary slip on the flow of a fluid through a microchannel, and to study the interaction force acting on the particles by the fluid and its resulting particle motion for various cases of particle initial velocity.

From the numerical results obtained, it can be concluded that the boundary slip between the fluid and the solid surface has very significant effect on the cross-section profile of the fluid velocity and the flow rate as well as the drag force acting on the immersed particles in the fluid.

It is also found that the inlet pressure required for driving the particles and fluid to flow with a required velocity increases very significantly with the increase of the number of particles immersed in the fluid. The results also indicate that the particle velocity will tend to constant after certain time after being injected into the fluid regardless its initial velocity.
Chapter 5

Movement of Particles in Fluids under Magnetic Forces

5.1 General Overview

To control magnetic particles (say drugs) to move in fluids to the target site, an external magnetic field is applied to generate a magnetic force acting on the particle [see 25]. The force is related to the magnetic field by the equation:

\[
F_{\text{mag}} = \frac{1}{\mu_r} (\mathbf{M} \cdot \nabla) \mathbf{B},
\]

(5.1)

where \( \mu_r \) is the relative permeability of a magnetic material, \( \mathbf{M} = (M_x, M_y, M_z) \) is the magnetization of the particle and \( \mathbf{B} = (B_x, B_y, B_z) \) is the magnetic flux density.

In this chapter, we first present a mathematical model to describe the fluid-particle motion in fluids under magnetic forces. Then we apply the finite element method, based on the Arbitary Lagrangian-Eulerian approach, to solve the underlying boundary value problem. Finally we consider the control of particle movement in fluids.

The rest of the chapter is organized as follows. In section 5.2, the boundary value problem for the determination of the magnetic flux density are presented followed by the finite element method for the solution of the problem. In section 5.3, the equations for the particle motion in fluids under magnetic forces are presented. In section 5.4, a numerical investigation is undertaken on the control of particle movement in fluids under magnetic fields.
5.2 Modelling of Magnetic Fields

For static condition in stationary bodies, the magnetic flux density $B$ is governed by Maxwell’s equations:

\[
\begin{align*}
\nabla \cdot B &= 0, \\
\nabla \times H &= 0,
\end{align*}
\]

where the magnetic flux density $B$ and the magnetic field strength $H$ are related through the constitutive relation

\[
B = \mu_0 \mu_r H + B_r,
\]

in which $B_r = \mu_0 \mu_r M$ denotes a residual flux density, $\mu_0$ is the permeability in vacuum.

From the first equation of (5.2), the magnetic flux density can be determined from a vector potential by $B = \nabla \times A$ which identically satisfies the first equation of (5.2). Using the identity

\[
\nabla \times (\nabla \times A) = \nabla (\nabla \cdot A) - \Delta A,
\]

and the Coulomb gauge $\nabla \cdot A = 0$, the second equation of (5.2) takes the form

\[
\nabla \times (\mu_0^{-1} \mu_r^{-1} \nabla \times A - M) = 0, \forall x \in \Omega_{def}
\]

or

\[
\Delta A = -\nabla \times (\mu_0 \mu_r M),
\]

which is the vector-valued Poisson equation for the magnetic potential $A$.

The boundary condition on particle surfaces is assumed to be insulation, that is,

\[
\mathbf{n} \times A = 0,
\]

where $\mathbf{n}$ is the unit normal vector. Equations (5.4) and (5.5) form the boundary problem for the determination of the magnetic field.

For the convenience in formulation, we rewrite the governing equations for magnetic field in indicial notation as follows,

\[
A_{i,jj} + \left(\nabla \times (\mu_0 \mu_r M)\right)_i = 0.
\]
The field equations (5.6) are defined in $\Omega_{de}(t)$ with the boundary conditions (5.5). These equations and boundary conditions constitute a closed system of three partial differential equations in terms of three unknown functions $A_1, A_2, \text{ and } A_3$. The system, supplemented by the boundary conditions (5.5) can be solved numerically to yield the magnetic vector potential $A$.

The variational boundary value problem associated with the problem is as follows.

**VBVP:** Find $A_i \in H^1(\Omega_{de})$ in the deformed mesh system at each time instant such that the Dirichlet boundary condition (5.5) is satisfied and for all $\hat{A}_i \in H^1_{A_i}(\Omega_{de}) \equiv \{ \hat{A}_i \in H^1(\Omega_{de}) | \hat{A}_i = 0 \text{ on } \partial \Omega_{de,A_i} \}$

$$
\left(A_{i,j}, \hat{A}_i \right) + \left( (\nabla \times (\mu_0 \mu_r M))_i, \hat{A}_i \right) = 0,
$$

(5.7)

where $H^1(\Omega_{de})$ is the Sobolev space $W^{1,2}(\Omega_{de})$ with norm $\| \cdot \|_{1,2,\Omega_{de}}$ and the inner product $(\cdot, \cdot)$ is defined by

$$
(a, b) = \int_{\Omega_{de}} (ab) d\Omega.
$$

(5.8)

Using integration by parts and the arbitrary of $\hat{A}_i$, we have

$$
\left(A_{i,j}, \hat{A}_{i,j} \right) - \left( (\nabla \times (\mu_0 \mu_r M))_i, \hat{A}_i \right) = 0,
$$

(5.9)

Since the computations are conducted in the reference coordinates, $\Omega$, we need to transform equations (5.9) in the deformed coordinates to those equations in the reference coordinates. Through this, the inner product expression in (5.8) is re-defined as

$$
(a, b) = \int_{\Omega_{de}} (ab) d\Omega = \int_{\Omega} (ab) |J| \, d\Omega.
$$

(5.10)

In the same way as that in chapter 4, the derivatives of the unknown functions $A_i$ and the test functions $\hat{A}_i$ are determined by the following expressions:

$$
\varphi_{i,j} = \varphi_{i,X_k} X_{k,j},
$$

(5.11)

$$
\hat{\varphi}_{i,j} = \hat{\varphi}_{i,X_k} X_{k,j}.
$$

(5.12)
Using (5.11) and (5.12), equations (5.13) become

\[
\left( A_{i,j}, \hat{A}_{i,j} \right) - \left( \nabla \times (\mu_0 \mu_r M), \hat{A}_i \right) = 0, \quad (5.13)
\]

To find the solution of the above VBVP problem, the Bubnov-Galerkin finite element method is used. We choose an \( M \)-dimensional subspace \( H^h_A \subset H^1_A(\Omega_{def}) \) for \( A_i \) and the corresponding test function. Let \( \{ \phi^A_m \}_{m=1}^M \) be the basis functions of \( H^h \) which is an \( M \)-dimensional subspace of \( H^1 \). Then we have

\[
A_i(X, t) \approx (A_i)_h = \sum_{m=1}^M \phi^A_m(X)(A_i)_m(t), \quad (5.14)
\]

\[
\hat{A}_i(X, t) \approx (\hat{A}_i)_h = \sum_{m=1}^M \phi^A_m(X)(\hat{A}_i)_m(t). \quad (5.15)
\]

Substituting equations (5.14)–(5.15) into equations (5.13), and following the same procedure as in chapter 4, we can solve the boundary value problem for the magnetic vector potential \( A \) and consequently, the magnetic flux density.

### 5.3 Modelling of Particle Movement

The motion of particles in fluids under magnetic fields is governed by Newton’s second law. As in chapter 4, we neglect the gravitational force, and hence the equations of motion for the particles in fluids under magnetic fields are as follows:

\[
m_q \frac{\partial V_q}{\partial t} = F_v + F_q + F_{mag}, \quad q = 1, 2, 3..., Q
\]

\[
V_q|_{t=0} = 0,
\]

where \( F_v \) is the fluid drag force acting on particle \( q \) by the fluid and can be determined by

\[
F_v = - \int_{\partial \Omega} \sigma \cdot n dS,
\]

in which \( \sigma \) is the stress in fluid and can be determined by the methods as presented in chapter 4. \( F_q \) are interaction forces acting on particle \( q \) by other particles and wall, namely

\[
F_q = \sum_{p=1,p\neq q}^{Q} F_{q,p} + \sum_{w=1}^{2} F_{q,w}, \quad (5.17)
\]
in which

\[ F_{q,p} = \begin{cases} 0, & \text{for } d_{q,p} > R_q + R_p + \alpha \\ \frac{1}{\varepsilon_q} (X_q - X_p)(R_q + R_p + \alpha - d_{q,p})^2, & \text{for } d_{q,p} \leq R_q + R_p + \alpha \end{cases} \]

and

\[ F_{q,w} = \begin{cases} 0, & \text{for } d_{q,w} > 2R_q + \alpha \\ \frac{1}{\varepsilon_w} (X_q - X_w)(2R_q + \alpha - d_{q,w})^2, & \text{for } d_{q,w} \leq 2R_q + \alpha \end{cases} \]

where \( d_{q,p} \) denotes the distance between the centers of the \( q \)th and \( p \)th particles, \( d_{q,w} \) denotes the distance between the center of the \( q \)th particle and the imaginary particle on the other side of the wall, \( X_q \) and \( R_q \) are center and radius of the \( q \)th particle, \( \alpha \) is the force range, and \( \varepsilon_q \) and \( \varepsilon_w \) are small positive stiffness parameters.

\( F_{\text{mag}} \) are magnetic forces acting on particle \( q \) which can be determined from (5.1). Also in a two dimension case, the magnetic potential is assumed to have a nonzero component only in the direction perpendicular to the plane, i.e., \( \mathbf{A} = (0, 0, A_z) \). On \( \partial \Omega_q \) and \( \partial \Omega \), the magnetic potential is set to zero, that is, \( A_z = 0 \).

The magnetization \( \mathbf{M} = (M_x, M_y) \) for the magnetic source is given by \( M_x = 0, M_y = 5 \times 10^4 \text{A} \cdot \text{m}^{-1} \), and for the magnetic particles

\[ M_x = a \arctan \left( \frac{b}{\mu_0 \mu_r A_{z,y}} \right), \]
\[ M_y = a \arctan \left( -\frac{b}{\mu_0 \mu_r A_{z,x}} \right), \]

(5.20)

where \( a \) and \( b \) are two material parameters. Thus from (5.1), the magnetic force, \( \mathbf{F}_{\text{mag}} = (F_{\text{mag},x}, F_{\text{mag},y}) \), is given by

\[ F_{\text{mag},x} = \frac{1}{\mu_r} (M_x A_{z,gy} + M_y A_{z,gy}), \]
\[ F_{\text{mag},y} = \frac{1}{\mu_r} (-M_x A_{z,xx} - M_y A_{z,xy}). \]

(5.21)

The position \( X_q \) of the center of the \( q \)th particle can be determined by the equation:

\[ \frac{dX_q}{dt} = V_q, \quad q = 1, 2, 3, ..., Q \]
\[ X_q|_{t=0} = X_q^0. \]

(5.22)
5.4 A Numerical Study on the Control of Particle Movement

In this section, we numerically study the movement of particles in fluids in a small vessel with the surrounding materials as shown in Figure 5.1. The computation domain is a horizontal channel with height of 3.1 µm and length of 45 µm. The particles are circular with diameter of 0.25 µm. The fluid is assumed to flow into the channel with the speed of 0.7 mm/s from the left to the right. The fluid properties are typical of human blood with the viscosity $\mu$ of 0.002 Pa·s and the density of 1040 kg·m$^{-3}$. All particles are assumed to be solid particles with the same density of 1400 kg·m$^{-3}$. The relative permeability $\mu_r$ is $5 \times 10^3$ for the magnet particles and 0.99998 for the tissue in the blood vessel. The material parameters $a$ and $b$ are $1 \times 10^{-4}$ A·m and $3 \times 10^{-5}$ (A·m)$^{-1}$, respectively.

Many cases with different number of magnetic particles are investigated and here we present the results for the case with one, three, five, and nine particles. Figure 5.1 shows the finite element mesh for the case with three particles. This typical mesh consist of 2956 elements.
Figure 5.2: Velocity profiles and particle locations at various instants of time for the case with three magnetic particles, shown together with the contour lines of the magnetic potential $A_z$. 
Figure 5.3: Magnetic forces acting on the particles for the three particles case: (a) the $x$-component of the magnetic force; and (b) the $y$-component of the magnetic force. The solid-triangle line is for particle 1; the solid-box line is for particle 2; and the solid-plus line is for particle 3.
Figure 5.4: The location of the particles as a function of time for the three particles case: (a) the $x$-coordinate; (b) the $y$-coordinate. The solid-triangle line is for particle 1; the solid-box line is for particle 2; and the solid-plus line is for particle 3.
Figure 5.2 shows the velocity profiles and the particle locations at various instants of time for the case with three magnetic particles together with the contour lines of the magnetic potential $A_z$. Figure 5.3 shows the magnetic forces acting on the particles, while Figure 5.4 shows the locations of the particles as a function of time. The results show that the external magnetic source generates a magnetic field leading to magnetic forces acting on the magnetic particles. Under these magnetic forces and the fluid drag forces, the particle moves gradually toward the location of the magnetic source.

5.5 Concluding Remarks

A mathematical model has been constructed to simulate the fluid-particle flow under magnetic fields, and has been applied to study the blood flow and the magnetic particle flow in a capillary vessel with an external magnetic source. From the results, it can be concluded that the proposed model and technique are capable of simulating the particle flow in fluid under magnetic forces. The particles move toward the magnetic source. By changing the location of the magnetic source, one can control the particles to move to the desired target sites.
Chapter 6

Conclusions

6.1 Summary

This project focuses on the development of mathematical models, analytical and numerical methods for the study of three microflow problems including the transient flow of fluids through micro-annulus with boundary slip, the particle-fluid flow in microchannels, and the particle motion in fluids under magnetic forces. The results and conclusions gained from the research are summarized below under three aspects.

(1) A mathematical model has been developed to study the pressure gradient driving transient flow of fluid through micro-annulus with boundary slip and the exact solution for the problem has been derived. Based on the analytical expressions of the solutions, we analyse the influence of the slip parameter $l$ and the geometry of the cross-section on the flow rate of fluid through the annual. The study shows that

(i) The influence of boundary slip on the flow through the annual is different for different types of pressure gradients. For flows driven by a constant pressure gradient, the flow rate always increases with the slip parameter $l$ and achieves a linear increase rate for large $l$ value; while, for the flows driven by the wave form pressure gradient with high frequency, the flow rate initially increases significantly as $l$ increases from zero but tends to a constant when $l$ becomes sufficiently large.
(ii) To achieve a fixed value of flow rate, one could have different \((k, l)\) designs. The exact solutions obtained, together with the contour plots of the solutions, provide a tool for engineers and scientists to determine the proper \((k, l)\) values.

(2) A mathematical model and numerical technique, based on the Arbitrary Eulerian-Lagrangian approach and the finite element method, has been developed for simulating the particle-fluid flow in microchannels. Based on the model and the numerical technique, various investigations have been carried out and the investigations show that

(i) The boundary slip between the fluid and the solid surface has very significant effect on velocity profile of the fluid and the flow rate as well as the drag force acting on the immersed particles in the fluid. For \(l = 0\) corresponding to the no-slip case, the velocity on the channel cross-section is basically parabolic except for the region close to the particles. As the slip parameter \(l\) increases, the velocity profile tends to uniform on the channel cross-section. Also, as \(l\) increased, the flow rate of fluid through the channel and the drag force acting on the immersed particles increase significantly.

(ii) The inlet pressure required for driving the particles and fluid to flow with a required velocity increases very significantly with the increase of the number of particles immersed in the fluid. Also, the particle velocity in fluid will tend to constant after certain time after being injected to the fluid regardless its initial velocity.

(3) A mathematical model and numerical technique has been established to study the motion of magnetic particles in fluid under magnetic fields generated by external magnetic sources. The model has been applied to study the blood flow and the movement of magnetic particles in the blood in a capillary vessel under an external magnetic source. The results show that the finite element based numerical technique is capable of simulating the particle flow in fluid under an external magnetic field. It is shown that the magnetic particles can
be controlled to move to the targeted site by changing the location of the external magnetic source.

### 6.2 Further Research

From the study, it can be concluded that mathematical modelling and numerical investigation can lead to a deeper understanding of the behaviour of fluid flow and particle flow in fluid in a micro-channel. Further work include: (a) study the fluid-particle flow in three dimension; (b) model blood flow at micro and/or nano scale based on non-Newtonian models.
Bibliography


Every reasonable effort has been made to acknowledge the owners of copyright material. I would be pleased to hear from any copyright owner who has been omitted or incorrectly acknowledged.