3 week project

Theory and Simulation of a Microfluidic instability Generator

Martin Heller, s001396
Søren Jacobsen, s991470
Torben Rasmussen, s991298

Supervisors: Henrik Bruus and Mads Jakob Jensen

MIC – Institute of Micro and Nanotechnology
Technical University of Denmark

21st January 2004
## Contents

List of figures \hspace{1cm} v  
List of tables \hspace{1cm} vii  
List of symbols \hspace{1cm} ix  
1 Introduction \hspace{1cm} 1  
2 Theory \hspace{1cm} 3  
\hspace{1cm} 2.1 Governing equations \hspace{1cm} 3  
\hspace{1cm} \hspace{1cm} 2.1.1 Continuity equation \hspace{1cm} 3  
\hspace{1cm} \hspace{1cm} 2.1.2 Navier-Stokes equation \hspace{1cm} 4  
\hspace{1cm} \hspace{1cm} 2.1.3 Surface tension and the Young-Laplace equation \hspace{1cm} 4  
\hspace{1cm} 2.2 Dimensionless numbers \hspace{1cm} 7  
\hspace{1cm} \hspace{1cm} 2.2.1 Reynolds number \hspace{1cm} 7  
\hspace{1cm} \hspace{1cm} 2.2.2 Bond number \hspace{1cm} 8  
3 Numerical methods \hspace{1cm} 9  
3.1 Numerical Simulation \hspace{1cm} 9  
4 Flows \hspace{1cm} 13  
\hspace{1cm} 4.1 Poiseuille flow with two different fluids \hspace{1cm} 13  
\hspace{1cm} 4.2 Transient time \hspace{1cm} 15  
\hspace{1cm} \hspace{1cm} 4.2.1 Circular pipe \hspace{1cm} 15  
\hspace{1cm} \hspace{1cm} 4.2.2 Parallel plates \hspace{1cm} 19  
\hspace{1cm} 4.3 Dispersion relation \hspace{1cm} 23  
\hspace{1cm} 4.4 Simulation results \hspace{1cm} 26  
\hspace{1cm} \hspace{1cm} 4.4.1 Summary \hspace{1cm} 29  
5 Conclusion \hspace{1cm} 31  
A Maple code \hspace{1cm} 35  
A.1 Two fluids \hspace{1cm} 35  
A.2 Cylindrical pipe \hspace{1cm} 35  
A.3 Parallel plates \hspace{1cm} 37
List of Figures

1.1 The system configuration is consisting of a microchannel with a piston in the side-wall. The piston is used to perturb the flow. ........................................ 1

2.1 Molecules in the bulk of a liquid binds together to minimise energy. Hence a dangling bond of a surface parcel gives rise to surface tension. ................................. 4

2.2 A pressure driven expansion of a surface balanced by the surface tension. .... 5

2.3 Channel, consisting of two infinite parallel plates. ................................. 6

2.4 Pressure difference across an interface. ................................. 7

3.1 This is the grid generated by CDF-GEOM. It is made by connecting the nodes placed along the lines, which connects the corners of every geometric piece of the whole figure ........................................ 10

3.2 A picture from CDF-VIEW. ................................ 10

4.1 Pressure driven flow of two liquids. ........................................ 13

4.2 The velocity profile of a Poiseuille flow with two fluids with different viscosities . 15

4.3 Velocity profile at different times. ........................................ 16

4.4 Plots of $v^\infty_0 (r,0)$. ........................................ 18

4.5 Two infinite parallel plates. ........................................ 19

4.6 Plot of the normalised velocity profile at different times $T = \frac{\lambda_0^{-2}(n/8)^3}{n=0,1,...,13}$. 22

4.7 The normalised theoretical velocity at $z = 0$ as a function of time. The velocity is approximately maximal ($v = 0.997v_{max}$) after $6\lambda_0^{-2} \approx 6.08$ ms. This is in excellent agreement with the simulation result. ........................................ 22

4.8 A small perturbation of the interface between two fluids. ................. 23

4.9 The dispersion relation. ........................................ 26

4.10 Plot showing the amplitude of the variation in density at the interface between the two fluids. Increasing amplitude suggest instable solution. CFD-ACE is unable to give the position of the interface. Instead the output is the average density of a computational cell. This number could be translated into a length scale by a linear translation. The oscillations will therefore be the same as shown in this plot. .... 28

4.11 Plot showing the amplitude of the variation in density at the interface between the two fluids. Increasing amplitude suggest instable solution. "Instable_4" has $\rho_1/\rho_2 = \frac{1}{2}$ and "Instable_5" has $\rho_1/\rho_2 = 2$. ........................................ 29

4.12 Plot showing the amplitude of the variation in density at the interface between the two fluids. The frequency of the piston is the only parameter being changed. Tendency lines are exponential approximations. ................................. 30
List of Tables

2.1 Parameters for the theoretic calculation. ........................................ 6
2.2 Parameters for the numerical simulation. ........................................ 6

4.1 Roots of $J_0$ and corresponding characteristic time constants. ........ 19
4.2 The first five time constants of the velocity summation for the case of two infinitely wide parallel plates. As it is seen the 0th time constant is a factor of 10 larger than the next one. This means that this characteristic time is the most dominant in the transient time. ....................................................... 21
4.3 Parameters used to simulate the instable plot shown in Figure 4.10. .... 27
4.4 Parameters used to simulate the instable plot shown in Figure 4.11. ..... 28
4.5 Parameters used to simulate the instable plot shown in Figure 4.12. ..... 29
# List of symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\rho)</td>
<td>Mass density</td>
<td>kg m(^{-3})</td>
</tr>
<tr>
<td>(\mathbf{x})</td>
<td>Position vector</td>
<td>m</td>
</tr>
<tr>
<td>(\mathbf{y})</td>
<td>Position vector</td>
<td>m</td>
</tr>
<tr>
<td>(\mathbf{z})</td>
<td>Position vector</td>
<td>m</td>
</tr>
<tr>
<td>(\mathbf{r})</td>
<td>Position vector</td>
<td>m</td>
</tr>
<tr>
<td>(\mathbf{v})</td>
<td>Velocity vector</td>
<td>m s(^{-1})</td>
</tr>
<tr>
<td>(\mathbf{v}, \mathbf{u})</td>
<td>Velocity vector</td>
<td>m s(^{-1})</td>
</tr>
<tr>
<td>(\mathbf{f})</td>
<td>Body force density</td>
<td>N m(^{-3})</td>
</tr>
<tr>
<td>(g)</td>
<td>Gravity</td>
<td>N kg(^{-1})</td>
</tr>
<tr>
<td>(\mathbf{n})</td>
<td>Surface outward normal vector</td>
<td></td>
</tr>
<tr>
<td>(p)</td>
<td>Pressure</td>
<td>N m(^{-2})</td>
</tr>
<tr>
<td>(\eta)</td>
<td>Dynamic viscosity</td>
<td>Pa s m(^{3}) kg(^{-1})</td>
</tr>
<tr>
<td>(\nu)</td>
<td>Kinematic viscosity</td>
<td>m(^{-2})</td>
</tr>
<tr>
<td>(J)</td>
<td>Surface current</td>
<td>J m(^{-2})</td>
</tr>
<tr>
<td>(\sigma)</td>
<td>Surface tension</td>
<td></td>
</tr>
<tr>
<td>(N)</td>
<td>Number of particles</td>
<td></td>
</tr>
<tr>
<td>(L, H, h, a)</td>
<td>Systems dimensions</td>
<td>m</td>
</tr>
<tr>
<td>(V)</td>
<td>Volume</td>
<td>m(^3)</td>
</tr>
<tr>
<td>(S, A)</td>
<td>Surface area</td>
<td>m(^2)</td>
</tr>
<tr>
<td>(\tau, \lambda^2)</td>
<td>Time constants</td>
<td>s</td>
</tr>
<tr>
<td>(\phi)</td>
<td>Potential function</td>
<td>m(^2) s(^{-1})</td>
</tr>
<tr>
<td>(k)</td>
<td>Wave number</td>
<td>m(^{-1})</td>
</tr>
<tr>
<td>(\omega)</td>
<td>Angular frequency</td>
<td>Hz</td>
</tr>
<tr>
<td>(\zeta)</td>
<td>Perturbation</td>
<td>m</td>
</tr>
<tr>
<td>(t)</td>
<td>Time</td>
<td>s</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

The physics of microfluidic systems is very different from the physics of macroscopic systems. Because of downscaling surface phenomena become much more dominant than volume phenomena. Moreover, flows in Microsystems are dominated by viscous effects, i.e., low Reynolds numbers. This results in laminar flow patterns. However, preferable in some situations it can be difficult to mix fluids streaming alongside each other. Mixing in microfluidic systems is mainly due to diffusion through the liquid-liquid interface, which is a slow process. Therefore, it would be of interest to find a method to speed this process up. One way to go is to perturb the liquid-liquid interface in order to obtain a larger surface for the particles to diffuse through.

If perturbation of the interface is to prove to be a usable solution the generated oscillations must be instable, i.e. grow in time and not die out. The investigation of the stability of surface perturbation in microfluidic flows will be the subject for this 3 week project.

We derive analytical solutions to various microfluidic configurations and compare the results with numerical solutions obtained from the simulation program CFD-ACE. This is done in order to gain confidence in the program and to obtain reliable solutions to the modeled problems.

We then simulate the pressure driven flow of two immiscible incompressible liquids perturbed by a piston in the channel wall. The simulations show that such perturbations are unstable for parameter values typical of microfluidic systems.

![Figure 1.1](image-url)  
**Figure 1.1:** The system configuration is consisting of a microchannel with a piston in the sidewall. The piston is used to perturb the flow.
Chapter 2

Theory

The goal of this project is to examine the flow of two different fluids streaming in a microchannel. Since flows in the micrometer regime is most often laminar two such liquids will only mix by diffusion through the interface. This is a relatively slow process. Hence it would be nice to speed this process up. One way to achieve this is by perturbing the interface so that it will be longer and allow for more diffusion. In the following sections we will derive some of the governing equations of fluidic dynamics.

2.1 Governing equations

2.1.1 Continuity equation

Considering a volume containing a given number of particles, resulting in a time dependant particle density within the volume $\rho(r, t)$, we investigate a particle flow out of the volume.

If $dN$ particles leave the system in the time $dt$, we find that this equals a net flux of particles out of a surface surrounding the volume

$$\frac{dN}{dt} = - \int_S J \cdot da,$$

(2.1)

This, however, can also be written as a change in density

$$\frac{dN}{dt} = \frac{\partial}{\partial t} \int_V \rho(r, t) d^3r$$

(2.2)

yielding

$$\frac{\partial}{\partial t} \int_V \rho(r, t) d^3r = - \int_S J \cdot da$$

(2.3)

$$= - \int_V \nabla \cdot J d^3r,$$

(2.4)

where we have used Gauss theorem. Since both integrals are independent of the shape surrounding the volume, we can write

$$\int_V \frac{\partial}{\partial t} \rho(r, t) d^3r = - \int_V \nabla \cdot J d^3r \Rightarrow$$

$$\frac{\partial}{\partial t} \rho(r, t) = - \nabla \cdot J$$

(2.6)
2.1. GOVERNING EQUATIONS

CHAPTER 2. THEORY

Figure 2.1: Molecules in the bulk of a liquid binds together to minimise energy. Hence a dangling bond of a surface parcel gives rise to surface tension.

which is known as the continuity equation. The current density \( J \) can be written as the product of the particle velocity and the density, giving us the alternative formulation

\[
\frac{\partial \rho(r,t)}{\partial t} + \nabla \cdot (\rho(r,t)v(r,t)) = 0
\]  

(2.7)

In this report we will only be using incompressible fluids, which means that \( \rho(r,t) \) is constant and that the continuity equation (2.7) reduces to

\[
\nabla \cdot (\rho v(r,t)) = 0
\]  

(2.8)

2.1.2 Navier-Stokes equation

The Navier-Stokes equation is Newton’s second law applied on a fluidic volume element (in this case incompressible). When considering fluid motion, it is no longer sufficient to consider spacial coordinates as time-independent as this would yield a velocity field for the entire fluid instead of a velocity field for an individual fluid particle. This means that the ordinary Newtonian equation of motion has a right hand side that reads

\[
\rho \left( \frac{dv(x(t),y(t),z(t),t)}{dt} \right) = \rho \left( \frac{\partial v}{\partial t} + (v \cdot \nabla) v \right).
\]

(2.9)

For a fluidic system the equation of motion (Navier-Stokes equation) then reads

\[
\rho \left( \frac{\partial v}{\partial t} + (v \cdot \nabla) v \right) = -\nabla p + \eta \nabla^2 v + f,
\]

(2.10)

where \( f \) denotes external forces acting on the system (body forces). These forces could, e.g., be originating from electrical fields or gravity. In order to neglect the gravitational term, we need to consider the so-called Bond number (see section 2.2.2). The additional term on the left \( (u \cdot \nabla) u \) is called the inertial term and can in some specific cases be neglected (see section 2.2.1).

2.1.3 Surface tension and the Young-Laplace equation

Surface tension is defined as the energy per unit area of an interface between two phases \([2]\). If we consider such an interface, the molecules positioned at the interface will have less neighboring atoms compared to the ones in the bulk (see Figure 2.1), resulting in a net inward force acting on the interface molecules. These molecules will hence have a higher free energy than those in the interior and as the system tend to minimise the energy, surface tension arise. The surface tension is therefore the energy per unit area that tends to minimise the area by contracting the surface \([2]\).
Figure 2.2: A pressure driven expansion of a surface balanced by the surface tension.

Considering an interface between two phases, the curvature of the interface will give rise to a pressure difference across the interface (see figure 2.2). This pressure difference can be found by considering the work needed to expand the surface. Characterising the surface by its radii of curvature $R_1$ and $R_2$ respectively, a small expansion of the radii with $\delta z$ will increase the surface area. The change in area can hence be found

$$R_1 \rightarrow R_1 + \delta z \quad \text{or} \quad 1 \rightarrow 1 + \frac{\delta z}{R_1}$$  \hspace{1cm} (2.11)

$$R_2 \rightarrow R_2 + \delta z \quad \text{or} \quad 1 \rightarrow 1 + \frac{\delta z}{R_2}$$  \hspace{1cm} (2.12)

$$A \rightarrow A + \delta A \quad \text{or} \quad A \rightarrow A + \frac{\delta z}{R_1 R_2}.$$  \hspace{1cm} (2.13)

The final expression can be expanded into

$$A + \delta A = A \left( 1 + \frac{\delta z}{R_1} + \frac{\delta z}{R_2} + \frac{\delta z^2}{R_1 R_2} \right).$$  \hspace{1cm} (2.14)

If we neglect the second order terms (as $\delta z$ is small) we get

$$\delta A \approx A \left( \frac{\delta z}{R_1} + \frac{\delta z}{R_2} \right).$$  \hspace{1cm} (2.15)

The work done by the surface as it stretches is then given by

$$\delta E_{\text{surface}} = \sigma \delta A = \sigma A \delta z \left( \frac{1}{R_1} + \frac{1}{R_2} \right),$$  \hspace{1cm} (2.16)

where $\sigma$ is the surface tension. This work, in equilibrium, has to be balanced by the work originating from the pressure difference

$$\delta E_{\text{pressure}} = -\Delta p A \delta z$$  \hspace{1cm} (2.17)

finally yielding the Young-Laplace equation

$$\sigma A \delta z \left( \frac{1}{R_1} + \frac{1}{R_2} \right) = \Delta p A \delta z \iff$$

$$\Delta p = \sigma \left( \frac{1}{R_1} + \frac{1}{R_2} \right).$$  \hspace{1cm} (2.18)
In this project we want to investigate the agreement between theory and numerical calculations, using the finite element method. The Young-Laplace equation is a good place to start as it is a simple problem with two different fluids. If we, furthermore, assume confinement of these fluids by two infinite parallel plates, the radius of curvature in one direction becomes infinite. Looking at Figure 2.3 we can express the radius of curvature by the dimensions of the system and the contact angle $\theta$. $2a$ is the height of the channel. We can then express the radius of curvature as $R = \frac{a}{\cos(\theta)}$.

Figure 2.3: Channel, consisting of two infinite parallel plates, containing two different fluids. Geometric interpretation of the contact angle expressed by the radius of curvature and the channel height.

Simplifying (2.19) into

$$\Delta p = \sigma \left( \frac{1}{R} \right) = \frac{\sigma \cos(\theta)}{a}.$$  

Using the following parameters for the theoretic calculation

<table>
<thead>
<tr>
<th>$\sigma$ [J/m$^2$]</th>
<th>$\theta$ [$^\circ$]</th>
<th>$a$ [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.018</td>
<td>85</td>
<td>50 $\times$ 10$^{-6}$</td>
</tr>
</tbody>
</table>

Table 2.1: Parameters for the theoretic calculation.

which correspond to the interface between water and olive oil, we get the following pressure difference

$$\Delta p = \left( \frac{0.018 \text{ J/m}^2 \times \cos(85^\circ)}{50 \times 10^{-6} \text{ m}} \right) = 31.376 \text{ Pa}.$$  

The numerical simulation needs a few more parameters describing the two fluids, which can be found in the table below

<table>
<thead>
<tr>
<th>$\sigma$ [J/m$^2$]</th>
<th>$\theta$ [$^\circ$]</th>
<th>$\eta_1$ [Pa s]</th>
<th>$\eta_2$ [Pa s]</th>
<th>$\rho_1$ [kg/m$^3$]</th>
<th>$\rho_2$ [kg/m$^3$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.018</td>
<td>85</td>
<td>10$^{-3}$</td>
<td>8.4 $\times$ 10$^{-2}$</td>
<td>1000</td>
<td>920</td>
</tr>
</tbody>
</table>

Table 2.2: Parameters for the numerical simulation.

After defining the geometry shown in Figure 2.3 with a channel height of $2a = 100 \mu$m and assigning the parameters above to the system, the simulation yields a plot showing the pressure difference across the interface (see Figure 2.4). The plot shows a pressure difference of 31.4 Pa, which is in good agreement with theory.
2.2 Dimensionless numbers

In order to characterise a fluidic system, several dimensionless numbers have been assigned describing various properties of the system. In this report we will only consider two of these numbers: Reynolds number relating inertial forces to viscous forces and the Bond number that relates gravitational forces to surface tensional force.

2.2.1 Reynolds number

Looking at the Navier-Stokes equation (2.10), we find the inertial term on the left hand side and the viscous term is identified as the second term on the right hand side. Setting up an equality between these two terms, we get

\[(v \cdot \nabla) v = \eta \nabla^2 v.\]  

(2.21)

Approximating the velocity field to a characteristic velocity \(u\) valid over a characteristic length \(L\) (this will be determined by the shortest length appearing in the system), we can make the substitution

\[\nabla \cdot v \approx \frac{u}{L},\]  

(2.22)

and

\[\nabla^2 v \approx \frac{u}{L^2}.\]  

(2.23)

Combining the above with (2.21) we get the approximation

\[Re = \frac{\rho (v \cdot \nabla) v}{|\eta \nabla^2 v|} \approx \frac{\rho u}{\eta L} = \frac{vL\rho}{\eta},\]  

(2.24)

(Reynolds number)
The higher the magnitude of the Reynolds number, the more dominant the inertial term will be. As it can be seen, the number is linearly dependant on the length scale, which means that we can automatically lower the inertial domination by decreasing the dimensions.

Reynolds number is divided into three regions:

**Laminar flow regime.** $Re < 2300$. Typically $Re$ is in the order of \( \frac{10^5 \text{kg/m}^3 \times 10^{-9} \text{m/s} \times 10^{-5} \text{m}}{10^{-5} \text{Pa} \cdot \text{s}} \approx 10^{-2} \) in microfluidic systems allowing the inertial term of (2.10) to be neglected. Steady flows dominate this regime like the Poiseuille flow.

**Transient flow regime.** $2300 < Re < 4000$. A mixture of both laminar and turbulent flow, with turbulent flow near the center of the flow channel and laminar flow at the edges.

**Turbulent flow regime.** $Re > 4000$. Chaotic movement of the fluid where vortices and waves make the flow unpredictable.

2.2.2 Bond number

The Bond number relates the gravitational force to the surface tensional force. This is an important number in trying to simplify the Navier-Stokes equation by neglecting body forces.

\[
Bo \approx \frac{\left| \rho L g \right|}{\sigma} = \frac{\rho g L^2}{\sigma} \quad \text{(Bond number)}
\]

In a typical micro fluidic systems this number is \( \frac{10^5 \text{kg/m}^3 \times 9.82 \text{kg m/s}^2 \times (10^{-5} \text{m})^2}{0.018 \text{ J/m}^2} \approx 0.05 \ll 1 \) allowing the gravitational term in the body forces to be neglected.
Chapter 3

Numerical methods

3.1 Numerical Simulation

The numerical simulations carried out in this project are done with CFD-ACE+, which is a commercial simulation program. The program is capable of numerically solving many of the partial differential equations known from the field of physics. It consists of different modules regarding fluid dynamics, but also some modules covering chemistry and biology are available. In our case only three modules are needed: flow, free surfaces, and grid deformation.

Flow: The Flow Module forms the basis of this program, it solves the mass- and momentum-conservation equations (or the Continuity Equation and Navier-Stokes). These are solved using a finite volume method. These equations provide the pressures and the velocities across the geometry.

Free Surfaces: This module also called VOF (Volume of Fluid) is used when two immiscible liquids are present, it also includes surface tension. VOF makes it possible to simulate systems containing two (or more) incompressible immiscible fluids. The simulation of the interface is done by linear approximation of the interface within each grid-cell also called PLIC, Piecewise Linear Interface Construction.

Grid Deformation: The Grid Deformation module is applied when a simulation of moving boundaries is needed, resulting in deformation of the grid. The boundaries can either be rotated or translated, the latter in our case, to create the sinusoidal perturbations on the interface of the liquids. This means that it is possible to create, e.g., a piston by choosing appropriate boundary conditions.

The program is divided into three small sub-programs:

- CFD-GEOM
- CFD-ACE(U)
- CFD-VIEW

CFD-GEOM is where the geometry of the problem is generated and the grid is defined. The geometry is made by creating points in a CAD like GUI, marking the corners and transitions of the boundaries in the design. Then connecting these points with lines and providing each line with a number of nodes which defines the resolution of the grid. The cells need to be square like, with
a ratio no higher than ten. Two opposing boundary lines has to have the same number of nodes. This makes it almost impossible to create a finer grid in certain areas of the geometry, which, e.g., could give a higher resolution without drastically increasing the calculation time. When all the grids are made, they are grouped and the final design is saved as a *.DTF file (see Figure 3.1).

![Figure 3.1](image)

**Figure 3.1:** This is the grid generated by CDF-GEOM. It is made by connecting the nodes placed along the lines, which connects the corners of every geometric piece of the whole figure

CDF-ACE(U) loads a *.DTF file containing the geometry and grid resolution. First the modules needed to describe and solve the problem are chosen (modules like the three listed above). When this is done the program will have the different parameters needed to define the problem displayed throughout the pages of the program. In our case we defined the two liquid involved, set the boundary conditions and initial conditions. This includes in- and outlets, pressure drop along the channel and the periodic translation of a wall (movement of the piston). It is important that the problem is set in the right orders of magnitudes for obvious reasons, this is done every time a new *.DTF file is loaded, as CDF-GEOM works in arbitrary units. Furthermore, there is the possibility of adjusting some of the parameters and methods used during the calculations. The length of the simulation and the timesteps between each calculation are defined. These parameters and the resolution of the grid determines the calculation time. Now, the problem is ready to be ‘submitted’ to the solver, one can continuously monitor the progress of the solver under ‘View Residuals’.

CDF-VIEW is where the result of the simulation may be seen. A line-figure of the geometry is presented and all the different parameters of interest can be displayed in this figure, e.g., the velocity and density can be shown (velocity as vectors and density as a color-gradient see Figure 3.2).

![Figure 3.2](image)

**Figure 3.2:** Here is a picture from CDF-VIEW of a channel containing two different fluids moving by laminar flow. The density is illustrated by the color gradient and the velocity by the vectors.

The Viewer may also plot these parameters along any given line. It can also make a plot of a parameter change in a given point as a function of time. This last plot will of course have its resolution defined by the timesteps of the simulation and have the total time span of the simulation, the lineplot will have the same resolution as the grid. There is an additional possibility to view the
CHAPTER 3. NUMERICAL METHODS  

3.1. NUMERICAL SIMULATION

Simulation as a movie, this, however, demands that the solver saves a file containing the data from all (or, e.g., every tenth, making the film more or less detailed) time step, which must be specified in CDF-ACE(U).

When we made these simulations the biggest problem was the calculation time. We got the best and most interesting simulations with a long channel, which obviously meant a lot of computational cells. Furthermore, the flow was generated by a pressure giving a transient time (see chap. 4.2) meaning that the results from the first time steps were useless.

Perturbation on the interface between the two liquids were created by a sinusoidal displacement of one of the walls in the channel (see Figure 1.1). The perturbation of the interface were difficult to monitor in CDF-VIEW. This is due to the fact that the solver does not save the exact position of the interface even though it is known at every time step of the calculation. The only thing which is saved is a number between one and zero giving the fraction of the two liquids in every cell.

It it possible to get a value for the magnitude of the amplitude of a perturbation by monitoring the change in density at a single point. Since we are studying a small perturbation of type \( z = z_0 + \zeta(x, t) \), a natural choice is to monitor the density at \( z = z_0 \) for a given \( x \). This is where the interface is at \( t = 0 \). As time evolves the perturbations in the channel will move the interface, shifting the density accordingly.

The software yields a linear density gradient instead of an absolute shift in density (as should be expected for insoluble fluids). Instead of getting discrete density changes, we get a continuous curve across the interface. Larger fluctuations in this curve would therefore mean larger displacement of the interface, even though we can not monitor the exact position, but we get a good approximation as the displacement and the density gradient should be linear dependent.
Chapter 4

Flows

In this chapter we will derive some analytical solutions to relevant microfluidic problems. This will enable us to verify the numerical solutions achieved using the CFD-ACE software. Furthermore, we investigate some relevant length- and timescales. We will start off with the derivation of the velocity profile for a steady state Poiseuille flow with two fluids of different viscosities. We then move on to investigations of the transient times for achieving such a flow profile in two different geometries. Finally, we derive a dispersion relation for a perturbed liquid-liquid interface.

4.1 Poiseuille flow with two different fluids

First we will study how two immiscible liquids move in a straight channel when they are not disturbed. That is, we will study the flow of two liquids streaming in steady state in the same channel. The flow is driven by the pressure drop $\Delta p$ over the channel length $L$. The reference pressure is $p_0$. Since the liquids are different they have different viscosities $\eta_1$ and $\eta_2$ and different velocities $v_1$ and $v_2$ respectively. The infinitely wide channel is confined by $0 < z < H$ and the liquid interface is positioned at $z = h$ (see figure 4.1).

Because of the geometry of the problem the velocities are given by $v_1 = (v_1(z), 0, 0)$ and $v_2 = (v_2(z), 0, 0)$. We are only considering the steady state case so the time dependent term in the Navier–Stokes equation disappears. This means that the Navier–Stokes equations for the two liquids reduces to

$$0 = -\nabla p + \eta_1 \nabla^2 v_1$$

and

$$0 = -\nabla p + \eta_2 \nabla^2 v_2 \quad (4.1)$$
4.1. POISEUILLE FLOW WITH TWO DIFFERENT FLUIDS

or in a more convenient notation

\[ 0 = \frac{\Delta p}{L} + \eta_1 \frac{d^2 v_1}{dz^2} \quad \text{and} \quad 0 = \frac{\Delta p}{L} + \eta_2 \frac{d^2 v_2}{dz^2}. \quad (4.2) \]

These are two second order differential equations. Thus we must require four boundary conditions in order to solve them. Assuming no-slip boundary conditions we can state that

\[ v_1(H) = 0 \quad \text{and} \quad v_2(0) = 0. \quad (4.3) \]

Furthermore we will assume that the two liquids have the same velocity at the interface, i.e.,

\[ v_1(h) = v_2(h) \quad (4.4) \]

and that the shear stress is the same on the two liquids on the interface

\[ \eta_1 \left. \frac{\partial v_1}{\partial z} \right|_{z=h} = \eta_2 \left. \frac{\partial v_2}{\partial z} \right|_{z=h}. \quad (4.5) \]

Integrating the equations (4.2) with respect to \( z \) solutions for \( v_1 \) and \( v_2 \) in the form of second order polynomials in \( z \) are obtained

\[ v_1 = -\frac{\Delta p}{2L\eta_1} z^2 + A_1 z + B_1 \quad (4.6) \]

and

\[ v_2 = -\frac{\Delta p}{2L\eta_2} z^2 + A_2 z + B_2. \quad (4.7) \]

Where \( A_1, A_2, B_1 \) and \( B_2 \) are integration constants to be determined from the boundary conditions. Applying the boundary conditions (4.3) \( B_2 \) is seen to be 0 and the equation

\[ 0 = -\frac{\Delta p}{2L\eta_1} H^2 + A_1 H + B_1 \Leftrightarrow B_1 = \frac{\Delta p}{2L\eta_1} H^2 - A_1 H \quad (4.8) \]

is found. Boundary condition (4.5) yields the relation

\[ A_2 = \frac{\eta_1}{\eta_2} A_1 \quad (4.9) \]

between \( A_1 \) and \( A_2 \). Using the last boundary condition (4.4) and eliminating \( A_2 \) and \( B_1 \) from the equations yields

\[ -\frac{\Delta p}{2L\eta_1} h^2 + A_1 h + \frac{\Delta p}{2L\eta_1} H^2 - A_1 H = -\frac{\Delta p}{2L\eta_2} h^2 + \frac{\eta_1}{\eta_2} A_1 h \Leftrightarrow \]

\[ A_1 = \frac{\Delta p}{2L\eta_1} \left[ h^2 \left( 1 - \frac{\eta_1}{\eta_2} \right) - H^2 \right] \frac{1}{h \left( 1 - \frac{\eta_1}{\eta_2} \right) - H}, \quad (4.10) \]

which can be inserted into the expressions for \( A_2 \) and \( B_1 \) respectively

\[ A_2 = \frac{\Delta p}{2L\eta_2} \left[ h^2 \left( 1 - \frac{\eta_1}{\eta_2} \right) - H^2 \right] \frac{1}{h \left( 1 - \frac{\eta_1}{\eta_2} \right) - H} \quad \text{and} \quad B_1 = \frac{\Delta p}{2L\eta_1} \left( H^2 - H \left[ h^2 \left( 1 - \frac{\eta_1}{\eta_2} \right) - H^2 \right] \right). \quad (4.11) \]
Thus the two velocity profiles becomes

\[ v_1(z) = -\frac{\Delta p}{2L \eta_1} \left[ z^2 - H^2 + (H - z) \frac{h^2 (1 - \frac{\eta_1}{\eta_2}) - H^2}{h (1 - \frac{\eta_1}{\eta_2}) - H} \right] \]  

(4.12)

and

\[ v_2(z) = -\frac{\Delta p}{2L \eta_2} \left[ z^2 - z \frac{h^2 (1 - \frac{\eta_1}{\eta_2}) - H^2}{h (1 - \frac{\eta_1}{\eta_2}) - H} \right]. \]  

(4.13)

A plot of the two velocity profiles is shown in Figure 4.2. Comparing this plot with the numerical simulation of a two fluid Poiseuille flow building up between two infinitely wide parallel plates (Figure 4.3) shows great resemblance of the flow profiles.

4.2 Transient time

In a real physical system a stationary flow profile is not obtained instantaneously. It is, therefore, interesting to investigate the timescales with which the profile is evolving. In the following sections we will study two different geometries: the circular pipe and two infinite parallel plates.

4.2.1 Circular pipe

In this section the transient time to obtain a Poiseuille flow in a circular pipe is studied. At time \( t = 0 \) a pressure difference of size \( \Delta p \) is induced over a circular pipe of radius \( a \) and length \( L \), where the liquid is still, \( v = 0 \).
Because the pressure drop is only in the \( x \)-direction the velocity is given by \( \mathbf{v} = \mathbf{v}(r, t) = (v_x(r, t), 0, 0) \). Putting this into the Navier–Stokes equation yields

\[
\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p \rho + \nu \nabla^2 \mathbf{v}. \tag{4.14}
\]

The inhomogeneous scalar differential equation (in cylindrical coordinates)

\[
\frac{\partial v_x}{\partial t} = \frac{\Delta p}{L \rho} + \nu \left( \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} \right) v_x \tag{4.15}
\]

is obtained.

Taking the symmetry of the problem into account and assuming no-slip boundary conditions, two boundary conditions in \( r \) can be found

\[
v_x(a, t) = 0 \quad \text{and} \quad \frac{\partial}{\partial r} v_x(0, t) = 0. \tag{4.16}
\]

Assuming that the system was at rest before introducing the pressure prop the initial condition

\[
v_x(r, 0) = 0 \tag{4.17}
\]

can be stated.

The solution to the steady state Poiseuille flow is known to be

\[
v_x(r, t) = \frac{\Delta p}{4 \eta L} (a^2 - r^2) \equiv v_x^\infty(r), \tag{4.18}
\]

which is a particular solution to (4.15). This can be verified by inserting into equation (4.15).
CHAPTER 4. FLOWS

4.2. TRANSIENT TIME

The complete solution to (4.15) can be written as the full solution \( u(r, t) \) to the homogene differential equation (4.20) plus the particular solution (4.18) to the inhomogen problem, i.e.,

\[
v(r, t) = u(r, t) + v_x^\infty(r).
\] (4.19)

To find the complete solution to the homogene differential equation

\[
\frac{\partial u}{\partial t} = \nu \left( \frac{\partial^2}{\partial r^2} + \frac{1}{r} \frac{\partial}{\partial r} \right) u
\] (4.20)
a product solution \( u(r, t) = R(r)T(t) \) is tried. Inserting into (4.20) yields

\[
RT' = \nu T \left( R'' + \frac{1}{r} R' \right).
\] (4.21)

Dividing with \( RT \) separates the variables

\[
\frac{T'}{T} = \nu \left( \frac{R''}{R} + \frac{1}{r} \frac{R'}{R} \right) = -\lambda.
\] (4.22)

Since the left hand side of this equation is only a function of \( t \) and the right hand side is only a function of \( r \), both sides must be constant (because \( t \) and \( r \) are independent). The boundary conditions for this equation can be found by using the conditions (4.16) and (4.17)

\[
u(a, t) = 0, \quad \frac{\partial u(0, t)}{\partial r} = 0 \quad \text{and} \quad u(r, 0) = -v_x^\infty(r).
\] (4.23)

The two separated equations are

\[
T' = -\lambda T \quad \text{and} \quad r^2 R'' + r R' + \frac{\lambda}{\nu} r^2 R = 0.
\] (4.24)

The equation in \( T \) is solved by a function of the form

\[
T = T_0 \exp(-\lambda t),
\] (4.25)

where \( \lambda \) is a positive constant and \( T_0 \) is the value of \( T \) at time \( t = 0 \). \( \lambda \) is positive because a negative value of \( \lambda \) would result in an unbounded solution. This is not physically possible.

The equation in \( R \) can be transformed by the following variable transformation

\[
x = \sqrt{\frac{\lambda}{\nu}} r \Rightarrow dx = \sqrt{\frac{\lambda}{\nu}} dr.
\] (4.26)

Inserting into the equation yields

\[
x^2 R'' + x R' + x^2 R = 0,
\] (4.27)

where \( R = R(x) \) and prime denotes differentiation with respect to \( x \). This is Bessel’s differential equation of order 0. The boundary conditions are

\[
R(a \sqrt{\frac{\lambda}{\nu}}) = 0 \quad \text{and} \quad R'(0) = 0.
\] (4.28)

The solution is the Bessel function of order 0; \( R(x) = J_0(x) \). Using the first boundary condition it is seen that

\[
a \sqrt{\frac{\lambda}{\nu}} = \lambda_k,
\] (4.29)
Using Eq. 27.112, the convergence of the summation is rather good. The velocities are normalised to Figure 4.4

where \( \tilde{\lambda}_k \) is the \( k \)'th root of \( J_0(x) \). Thus \( u \) must be given by the Fourier like sum

\[
u(r, t) = \sum_{k=1}^{\infty} T_k R_k = \sum_{k=1}^{\infty} T_0^k \exp(-\lambda_k t) J_0(\sqrt{\frac{\lambda_k}{\nu}} r).
\]

(4.30)

Using the initial condition it is found that

\[
u(r, 0) = \sum_{k=1}^{\infty} T_0^k J_0(\sqrt{\frac{\lambda_k}{\nu}} r) = -v_{x}^\infty(r)
\]

(4.31)

\[\int_0^1 \tilde{r} v_{x}^\infty(r) J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r}, \]

(4.32)

where the coordinate transformation \( \tilde{r} = \frac{r}{a} \) and the definition (4.29) has been used. Evaluating the integral using (4.18) yields

\[
\int_0^1 \tilde{r} v_{x}^\infty(r) J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r} = \frac{a^2 \Delta p}{4\eta L} \int_0^1 \tilde{r} (1 - \tilde{r}^2) J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r} = \frac{a^2 \Delta p}{4\eta L} \left[ \int_0^1 \tilde{r} J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r} - \int_0^1 \tilde{r}^3 J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r} \right].
\]

(4.33)

The integrals in the squared brackets are evaluated using [4, Eq. 27.74] and [4, Eq. 27.76] after applying the coordinate transformation \( x = \tilde{\lambda}_k \tilde{r} \)

\[
\int_0^1 \tilde{r} J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r} = \frac{1}{\tilde{\lambda}_k^2} \int_0^{\tilde{\lambda}_k} x J_0(x) dx - \frac{1}{\tilde{\lambda}_k^3} \int_0^{\tilde{\lambda}_k} x^3 J_0(x) dx = \frac{1}{\tilde{\lambda}_k^2} [x J_1(x)]_{0}^{\tilde{\lambda}_k} - \frac{1}{\tilde{\lambda}_k^3} \left[ x^3 J_1(x) + 2x^2 J_0(x) - 4x J_1(x) \right]_{0}^{\tilde{\lambda}_k}
\]

(4.34)

\[
\frac{4 J_1(\tilde{\lambda}_k)}{\lambda_k^3},
\]

Figure 4.4: Plots of the right hand side and the left hand side of equation (4.31) respectively reveals that the convergence of the summation is rather good. The velocities are normalised to \( \frac{\Delta p}{4\eta L} \).

Using [4, Eq. 27.112] \( T_0^k \) can be calculated as

\[
T_0^k = \frac{-2}{J^2(\lambda_k)} \int_0^1 \tilde{r} v_{x}^\infty(r) J_0(\tilde{\lambda}_k \tilde{r}) d\tilde{r},
\]

18
where it has been used that \( \tilde{\lambda}_k \) is root in \( J_0(x) \) and hence \( J_0(\tilde{\lambda}_k) = 0 \). The solution to the inhomogenous equation (4.15) is therefore

\[
v(r, t) = v_x^\infty(r) - \frac{2a^2\Delta p}{\eta L} \sum_{k=1}^{\infty} \frac{1}{\tilde{\lambda}_k^2 J_1(\tilde{\lambda}_k a)} J_0(\tilde{\lambda}_k a) e^{-\frac{r}{\tau_k}},
\]

(4.35)

where \( \tau_k = \frac{1}{\nu} \frac{a^2}{\tilde{\lambda}_k^2} \) is the characteristic time constant of the system.

<table>
<thead>
<tr>
<th>k</th>
<th>( \tilde{\lambda}_k )</th>
<th>( \tau_k/(10^{-6} \text{s}) )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.4048</td>
<td>432.29</td>
</tr>
<tr>
<td>2</td>
<td>5.5200</td>
<td>82.04</td>
</tr>
<tr>
<td>3</td>
<td>8.6537</td>
<td>33.38</td>
</tr>
<tr>
<td>4</td>
<td>11.7915</td>
<td>17.98</td>
</tr>
<tr>
<td>5</td>
<td>14.9309</td>
<td>11.21</td>
</tr>
</tbody>
</table>

Table 4.1: The first five roots \( \tilde{\lambda}_k \) of the Bessel function of first kind of order 0 and the corresponding characteristic time constants \( \tau_k \). In the calculation of \( \tau_k \) typical values of \( a \) and \( \nu \) has been used. That is \( a = 50 \mu \text{m} \) and \( \nu = 1 \times 10^{-6} \text{m}^2\text{s}^{-1} \).

As it is seen from Table 4.1 the magnitudes of the time constants drop rapidly. This means that only the first terms of the summation (4.35) are of importance due to the exponential dependance of \( \tau_k \). As seen from figure 4.4 the convergence is very good. The plot in figure 4.4(a) is made with 10 terms in the summation. This gives a very good correspondence between the analytical expression for \( v_x^\infty(r, 0) \) and the approximation. Figure 4.4(c) reveals that the error is less than 0.01 \( \frac{\Delta \rho g a^2}{\eta L} \) already when only five terms is used in the summation.

### 4.2.2 Parallel plates

When simulating flows using ACE(U) it is advantageous to restrict the problem at hand to two dimensions, because it saves valuable time. The problem with transient time in a circular pipe requires tree dimensional calculations and, furthermore, it would seem difficult to actually achieve circular features using known micro machining facilities.

Instead we could choose the problem with two parallel plates not bounded in the direction perpendicular to the pressure gradient (see Figure 4.5). As before the velocity profile is translation invariant along the x-axis giving us a velocity profile only dependant on z, with the only component \( v_x(z, t) \): \( v = (v_x(z, t), 0, 0) \).

Figure 4.5: Two infinite parallel plates with an applied pressure drop at times \( t = 0 \) and \( t = \infty \). With only one fluid.
4.2. TRANSIENT TIME

CHAPTER 4. FLOWS

Writing the Navier-Stokes equation for \( v_x(z,t) \) we get

\[
\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} = - \frac{\nabla p}{\rho} + \nu \nabla^2 \mathbf{v}.
\]

(4.36)

Neglecting the inertial term due to symmetry we finally achieve the inhomogeneous second order differential equation

\[
\frac{\partial v_x}{\partial t} = \frac{\Delta p}{\rho} + \nu \frac{\partial^2 v_x}{\partial z^2}.
\]

(4.37)

As was the case with the circular flow, we also know the velocity profile for such a problem when time goes to infinity, namely the steady state Poiseuille flow given by

\[
v_x(z, t) = \frac{\Delta p}{2 \eta L} \left(a^2 - z^2 \right) \equiv v_x^\infty(z).
\]

(4.38)

This profile becomes our first boundary condition. Moving on we must also demand

\[
v_x(\pm a, t) = 0
\]

(4.39)

and

\[
v_x(z, 0) = 0.
\]

(4.40)

The total solution to (4.37) is given by

\[
v_x(z, t) = v_f(z) + v_H(z, t),
\]

where \( v_f(z) = v_x^\infty(z) \) is a particular solution to the inhomogeneous problem. Looking at the homogeneous problem we get

\[
\frac{\partial v_H}{\partial t} = \nu \frac{\partial^2 v_H}{\partial z^2}
\]

(4.42)

with the boundary conditions

\[
v_H(-a, t) = v_H(a, t) = 0
\]

(4.43)

and since \( v_x(z, 0) = 0 \) we must also demand

\[
v_H(z, 0) = -v_x^\infty(z).
\]

(4.44)

Once again we try with a product solution in \( z \) and \( t \), writing

\[
v_H(z, t) = Z(z)T(t) \Rightarrow T'Z = \nu TZ''
\]

(4.45)

dividing with \( ZT \) separates the variables

\[
\frac{T}{T} = \nu \frac{Z''}{Z} = -\lambda^2, \forall \lambda > 0.
\]

(4.46)

Separating the solutions we first solve for the time dependance, which is an ordinary first order differential equation

\[
T' = -\lambda^2 T \Rightarrow T(t) = c_1 e^{-\lambda^2 t}, \lambda > 0,
\]

(4.47)
CHAPTER 4. FLOWS

4.2. TRANSIENT TIME

where we have discarded the positive solution as the function has to tend to zero as time increases. This ensures that (4.44) is fulfilled.

The \( z \)-dependence gives a second order homogenous differential equation

\[
Z'' = -\frac{\lambda^2}{\nu} Z \Rightarrow Z(z) = c_2 \cos \left( \frac{\lambda}{\sqrt{\nu}} z \right) + c_3 \sin \left( \frac{\lambda}{\sqrt{\nu}} z \right).
\]

In order to fulfil (4.43) we have to discard the sine solution and set \( c_3 = 0 \). This gives us

\[
v_H(z, t) = A e^{-\lambda^2 t} \cos \left( \frac{\lambda}{\sqrt{\nu}} z \right).
\]

Using (4.44) we find that

\[
\frac{\lambda a}{\sqrt{\nu}} = \left( \frac{\pi}{2} + n\pi \right) \Rightarrow \lambda_n = \frac{\sqrt{\nu}}{a} \left( \frac{\pi}{2} + n\pi \right).
\]

The \( n \)’th solution then reads

\[
v_H(z, t) = A_n e^{-\lambda^2 t} \cos \left( \frac{\lambda_n}{\sqrt{\nu}} z \right), \quad n \in N
\]

and using the final boundary condition (4.44) we get

\[
\Lambda_n \cos \left( \frac{\lambda_n}{\sqrt{\nu}} z \right) = -k_1 (a^2 - z^2) \Rightarrow A_n = -\frac{1}{a} \int_{-a}^{a} \cos \left( \frac{\lambda_n}{\sqrt{\nu}} z \right) k_1 (a^2 - z^2)dz
\]

\[
= -\frac{32}{\pi^3} \frac{a^2 (-1)^n k_1}{(1 + 6n + 12n^2 + 8n^3)}
\]

where \( k_1 = \frac{\Delta p}{2\eta L} \).

The final time dependant velocity profile then becomes

\[
v(z, t) = v_{x}^{\infty}(z) - \sum_{n=0}^{\infty} \frac{a^2 (-1)^n k_1}{\pi^3 (1 + 6n + 12n^2 + 8n^3)} e^{-\lambda^2 t} \cos \left( \frac{\lambda_n}{\sqrt{\nu}} z \right),
\]

where

\[
\lambda_n = \frac{\sqrt{\nu}}{a} \left( \frac{\pi}{2} + n\pi \right) \quad \text{and} \quad k_1 = \frac{\Delta p}{2\eta L}
\]

calculating the first five time constants we obtain the values tabulated in Table 4.2.

<table>
<thead>
<tr>
<th>( n )</th>
<th>0</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \lambda_n^{-2} ) /ms</td>
<td>1.0132</td>
<td>0.1125</td>
<td>0.0206</td>
<td>0.0125</td>
<td></td>
</tr>
</tbody>
</table>

**Table 4.2:** The first five time constants of the velocity summation for the case of two infinitely wide parallel plates. As it is seen the 0th time constant is a factor of 10 larger than the next one. This means that this characteristic time is the most dominant in the transient time.

As can be seen from Figure 4.6 the steady state velocity profile is reached within the time window of approximately 4 times the 0th time constant. This means that the flow profile does not change after 4 ms. The transient time is only depending on the kinematic viscosity of the fluid and
4.2. TRANSIENT TIME

Figure 4.6: Plot of the normalised velocity profile at different times $T = \lambda_0^{-2}\left(n/8\right)^3$, $n = 0, 1, \ldots, 13$.

Figure 4.7: The normalised theoretical velocity at $z = 0$ as a function of time. The velocity is approximately maximal ($v = 0.997v_{\text{max}}$) after $6\lambda_0^{-2} \approx 6.08$ ms. This is in excellent agreement with the simulation result.
chapter 4. flows

4.3. dispersion relation

the channel dimension. Every other system parameter is only affecting the maximum velocity, but not the time it takes to obtain it.

Simulating the Poiseuille flow between two infinite parallel plates yields the time development in the maximal velocity shown in Figure 4.7(b). The maximal velocity can be calculated as

\[ v_{\text{max}} = \frac{\Delta p a^2}{2\eta L} = \frac{2000 \text{ Pa} \times (50 \mu \text{m})^2}{1000 \mu \text{m} \times 0.001 \text{ Pa s}} = 2.5 \text{ m s}^{-1} \]  \hfill (4.56)

As can be seen from the plot this velocity is obtained around 6 ms where the curve is very flat. This in excellent correspondence with the theoretically calculated transient time (see Figure 4.6).

4.3 dispersion relation

In this section we consider what happens if a liquid–liquid interface is perturbed and derive a dispersion relation. We will examine the highly simplified situation where the interface is infinitely far away from any walls, and the interface is infinitely long. Furthermore, we will neglect viscosity because the viscosity term complicates the calculations greatly. It is not within the time limits of this 3 week project to go into this kind of details.

The two incompressible liquids are characterised by the constant densities \( \rho_1 \) and \( \rho_2 \). The interface is connected with the surface tension \( \sigma \) and is subjected to the small perturbation \( \zeta(x,t) \) in the \( z \)-direction. This is a 2D case. In the 3D case the perturbation would also be a function of \( y \).

Figure 4.8: A small perturbation of the interface between two incompressible immiscible fluids of different viscosities and densities. The interface between the two fluids is characterised by the surface tension \( \sigma \).

The Navier-Stokes equation is

\[ \rho_i \partial_t \mathbf{v} + \rho_i (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p_i + \eta_i \nabla^2 \mathbf{v}_i + \rho_i \mathbf{g}. \]  \hfill (4.57)

Canceling the viscous term reduces the Navier-Stokes equation to

\[ \rho_i \partial_t \mathbf{v} + \rho_i (\mathbf{v} \cdot \nabla) \mathbf{v} = -\nabla p_i + \rho_i \mathbf{g}. \]  \hfill (4.58)

Because the fluids are assumed to be incompressible the continuity equation is valid, i.e.,

\[ \nabla \cdot \mathbf{v} = 0. \]  \hfill (4.59)

Assuming that the velocity field has no rotation we can use that

\[ \nabla \times \mathbf{v} = 0 \Rightarrow \mathbf{v} = \nabla \phi_i, \]  \hfill (4.60)

where \( \phi_i \) is a potential function. Obviously the unit of \( \phi \) is \( [\phi] = \text{m}^2\text{s}^{-1} \). Combining (4.59) and (4.60) yields a Laplace equation in \( \phi \)

\[ \nabla^2 \phi_i = 0. \]  \hfill (4.61)
Because the rotation of the velocity field is 0 the inertial term in the Navier-Stokes equation can be rewritten as
\[(\mathbf{v} \cdot \nabla)\mathbf{v} = \partial_j \left( \frac{1}{2} v_j^2 \right).\] (4.62)
The gravitational term can be rewritten as
\[\rho g = -\nabla (gz).\] (4.63)
Inserting (4.60), (4.62) and (4.63) in (4.58) and dividing with \(\rho_i\) yields
\[\nabla \left( \frac{1}{2} v_i^2 + gz + \frac{p_i}{\rho_i} + \partial_t \phi_i \right) = 0.\] (4.64)
Assuming \(\phi_i\) is plain waves of the form
\[\phi_i = a_i f(z)e^{i(kz-\omega t)},\] (4.65)
we can insert into equation (4.61) and find
\[\partial^2_z \phi_i = k^2 \phi_i \Rightarrow f(z) = e^{\pm kz}\] (4.66)
and thus
\[\phi_i = a_i e^{\pm kz} e^{i(kz-\omega t)}.\] (4.67)
Because we are only perturbing the interface we will assume that the velocity infinitely far away from the interface is zero at all times. This assumption leads to the boundary condition
\[v_i(z = \pm \infty) = 0.\] (4.68)
Using this boundary condition yields
\[f_2(z) = e^{-kz} \Rightarrow \phi_2(x, z, t) = a_2 e^{-kz} e^{i(kz-\omega t)}\] (4.69)
and
\[f_1(z) = e^{kz} \Rightarrow \phi_1(x, z, t) = a_1 e^{kz} e^{i(kz-\omega t)}.\] (4.70)
Now we will introduce a small perturbation \(\zeta(x, 0, t)\) of the interface. This enables us to find the \(z\)-component of the velocity of the interface
\[v_z|_{z=0} = \frac{D\zeta}{Dt} = \frac{\partial \zeta}{\partial t} + (\mathbf{v} \cdot \nabla)\zeta = \frac{\partial \zeta}{\partial t} + [(\mathbf{v}_0 + \mathbf{v}') \cdot \nabla] \zeta = \partial_t \zeta.\] (4.71)
The square brackets are 0 because \(v_0\) is zero and \(v'\) is second order in \(\zeta\). Therefore the equation reduces to
\[v_z = \partial_z \phi_1 = \partial_t \zeta, \quad z = 0.\] (4.72)
We will now assume that in the limit of \(z \to \infty\) the perturbation wave will tend to 0. Applying this assumption to (4.64) yields
\[\frac{1}{2} v_i^2 + gz + \frac{p_i}{\rho_i} + \partial_t \phi_i = 0\] (4.73)
and

\[ \frac{1}{2} v_1^2 + g z + \frac{p_1}{\rho_1} + \partial_t \phi_1 = 0. \]  

(4.74)

Multiplying these equations with \( \rho_2 \) and \( \rho_1 \) respectively and subtracting them we find

\[ \frac{1}{2} (v_2^2 \rho_2 - v_1^2 \rho_1) + zg(\rho_2 - \rho_1) + \sigma(\rho_2 - \rho_1) + \partial_t (\rho_2 \phi_2 - \rho_1 \phi_1) = 0. \]  

(4.75)

At \( z = 0 \) this equation reduces to

\[ \frac{1}{2} (v_2^2 \rho_2 - v_1^2 \rho_1) + \sigma \Delta p + \partial_t (\rho_2 \phi_2 - \rho_1 \phi_1) = 0. \]  

(4.76)

Using the Young-Laplace equation (that the pressure difference over a interface is the surface tension times the curvature of the interface (Equation (2.19))) and the fact that at \( z = 0 \) the slope of the perturbation wave must be the same from both sides of the interface: \( \partial_z \phi_1 \big|_{z=0} = \partial_z \phi_2 \big|_{z=0} \Rightarrow a_2 = -a_1 \) and hence \( \phi_2(x,0,t) = -\phi_1(x,0,t) \) the above equation reduces to

\[ \frac{1}{2} (v_2^2 \rho_2 - v_1^2 \rho_1) + \sigma \partial^2 \zeta \big|_{z=0} + (\rho_1 + \rho_2) \partial_t \phi_1 = 0. \]  

(4.77)

The time derivative of \( \phi_1 \) is found from (4.72) to be

\[ \partial_t \phi_1 = -i\omega \phi_1 = i\omega \frac{i\omega}{k} \zeta. \]  

(4.78)

Combining the above results in

\[ (\rho_2 - \rho_1) g \zeta - \sigma k^2 \zeta + (\rho_1 + \rho_2) \frac{\omega^2}{k} \zeta = 0, \]  

(4.79)

where it has been used that the terms containing the velocities are second order in \( \zeta \). Rearranging the equation gives us the dispersion relation for viscousless fluids with gravitation,

\[ \omega^2 = \frac{\sigma k^2}{\rho_1 + \rho_2} + \frac{\rho_2 - \rho_1}{\rho_1 + \rho_2} g k. \]  

(4.80)

In the gravitation free case the frequency is proportional to \( k^2 \). In order for the system to be unstable \( \omega^2 \) must be negative. This is not possible if neither gravitation or viscosity is taken into account. Hence, this dispersion relation can not be used to predict if the simulated systems are stable or unstable. Nevertheless the dispersion relation can be used to check that the wavelengths found in the simulation are of the correct order of magnitude.

The derived dispersion relation is of cause not containing the relevant physics of microfluidic systems. As discussed earlier gravity is not important in microsystems because it is neglectable in comparison with the surface forces (low Bond number). Furthermore, we have only considered the inviscid case in the derivation of the dispersion relation. This is obviously not a valid assumption in microfluidic systems where the Reynolds numbers are very small.

Basically the derived dispersion relation can only be used to check that the wavelengths are of the correct order of magnitudes compared to the driving frequency. It is not possible to examine the stability of a microsystem based on this dispersion relation. We can only discuss the concept of stability of a fluid-fluid interface qualitatively for microsystems. As can be seen from Figure 4.9, where three measured values of \( \omega \) and \( k \) has been cotted on the same graph as the derived expression
for the dispersion relation, the correspondence between theory and simulation is not very good. This was as expected, but the orders of magnitudes is correct.

All systems will tend towards a state that is energetically favorable. That is, the system will try to minimize its energy. In a microchannel containing two liquids with no flow it costs a lot of energy to obtain a surface (the interface between the liquids). This energy is much larger than the gravitational energy stored in the system. Hence the system will try to minimize the area of the interface (we are not taking into consideration if the channel surface is hydrophillic or hydrophobic etc.). In a microchannel the length of the channel is often several orders of magnitudes larger than the width and the height of the channel. Therefore, the situation where two liquids meet at an interface along the entire length of the channel is unstable because the liquids could rearrange in such a way, that the interfaces would be transversal to the channel. This would reduce the energy of the system, because only little gravitational energy would be gained compared with the drop in energy stored in the interface surface.

This quantitative explanation might explain the fact that we are only observing unstable perturbations of the two liquid Poiseuille flow in our simulations. This result is supported by the results of [3].

4.4 Simulation results

As demonstrated CFD-ACE yields results in good agreement with the expected theoretical expressions, we move on to the actual simulation of perturbed systems.

More thorough investigations of a perturbed double fluidic system [3, 5] with Poiseuille flow,
reveals that several factors are determining the behavior of the interface between the fluids. These
analysis were based on three vital dimensionless factors, namely the relation between:

- the densities of the two fluids \( \frac{\rho_1}{\rho_2} \).
- the viscosities of the two fluids \( \frac{\eta_1}{\eta_2} \).
- the height of each fluid in the \( z \)-direction \( \frac{h_1}{h_2} \).

It was shown that the system could be put in a state of instability, which would cause the interface
between the two fluids to curve in a manner that would not "die out" as a damped oscillation. Certain of the above mentioned relations would instead invoke a exponentially growing amplitude
of the interface, meaning that it should (in theory) be possible to elongate the interface rapidly
and hereby achieve, e.g., a mixture by diffusion of the two fluids within a shorter horizontal distance
than with a steady laminar unperturbed system.

References [3, 5] showed that it was possible to achieve instable solutions to the perturbed
system for a variety of relations between the fluid heights, including the relation \( \frac{h_1}{h_2} = 1 \), so we
choose to keep the height of each fluid equal.

All theoretical predictions were made with gravitational forces applied, but this setting did not
yield convergent results in the numeric solver, so this option was not used.

From our investigation of transient time, we know that a typical time to obtain maximum
velocity is around 6 milliseconds. This means that we should at least create simulations over this
time period. All simulations were performed with a duration above 10 orders of magnitude of the
transient time.

It was shown in [3] that if \( \frac{\rho_1}{\rho_2} > 1 \), \( \frac{\eta_1}{\eta_2} \geq 1 \) (and \( \frac{h_1}{h_2} = 1 \)) instable solutions are obtained. Using
these parameters in our simulation we set \( \rho_1 = 1000 \text{ kg/m}^3 \) and \( \rho_2 = 900 \text{ kg/m}^3 \). Figure 4.10 shows the plot of the amplitude as a function of the horizontal position in the channel. The solutions shows a growing amplitude as predicted by theory. "Instable_1" and "Instabale_2" are simulated
with surface tension \( \sigma_1 \), and "Instable_3" is simulated with \( \sigma_2 \) (see Table 4.3).

Changing the position of fluid 1 and 2, was also predicted to yield an instable solution (which
makes sense in our case as no gravitational forces are applied). Figure 4.10 shows this situation
(instable_2) and as it can be seen, the amplitude does increase.

<table>
<thead>
<tr>
<th>Piston width [μm]</th>
<th>Frequency [Hz]</th>
<th>Amplitude [μm]</th>
<th>( \sigma_1 ) [J/m²]</th>
<th>( \sigma_2 ) [J/m²]</th>
<th>Pressure [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>640</td>
<td>1000</td>
<td>3</td>
<td>0.0018</td>
<td>0.00018</td>
<td>3000</td>
</tr>
</tbody>
</table>

Table 4.3: Parameters used to simulate the instable plot shown in Figure 4.10.

In the next simulation we chose the following values for the relations: \( \frac{\rho_1}{\rho_2} = 2 \), \( \frac{\eta_1}{\eta_2} = \frac{1}{2} \), \( \frac{h_1}{h_2} = 0.43 \)
and \( \frac{h_1}{h_2} = 2 \). This means that the height of the top fluid is 30 μm and the height of the lower fluid
is 70 μm. The surface tension has been set to a more realistic value.

Figure 4.11 shows the plot of the amplitudes. As it can be seen, this did not change the
situation and both plots reveal instable tendencies. The tendency does, however, no longer follow
an analytical function such as the exponentially growing amplitude, but seems to have a kink along
the axis of propagation. No physical explanation seems reasonable for this behaviour and should
probably be assigned to a faulty numerical solution rather than the parameters.

Common in these simulations were the following parameters:
4.4. SIMULATION RESULTS

Figure 4.10: Plot showing the amplitude of the variation in density at the interface between the two fluids. Increasing amplitude suggest instable solution. CFD-ACE is unable to give the position of the interface. Instead the output is the average density of a computational cell. This number could be translated into a length scale by a linear translation. The oscillations will therefore be the same as shown in this plot.

<table>
<thead>
<tr>
<th>Piston width [µm]</th>
<th>Frequency [Hz]</th>
<th>Amplitude [µm]</th>
<th>σ [J/m²]</th>
<th>Pressure [Pa]</th>
</tr>
</thead>
<tbody>
<tr>
<td>640</td>
<td>1000</td>
<td>3</td>
<td>0.0725</td>
<td>2000</td>
</tr>
</tbody>
</table>

Table 4.4: Parameters used to simulate the instable plot shown in Figure 4.11.

The increase in surface tension, compared to the first simulation, was expected to yield a more stable solution, but a clear tendency can not be derived from Figure 4.11. Instability is still present, in agreement with [3].

The last simulation was conducted in an longer channel, as we wanted a clearer picture of how the perturbation evolved. This simulation required a lot of computational power, limiting the amount of results. Three simulations were carried out using the same parameters, only varying the frequency of the piston. As Figure 4.12 shows increasing amplitude, which can be approximated by a exponential function for 500 Hz and 1000 Hz. The plot of the amplitude for 250 Hz does not fit the curve. Large irregularities in amplitude appeared in this simulation, just like Figure 4.11. The cause of these irregularities is to this point unknown, but it would seem that the fault lies in the parameters used. A refinement of the grid might also resolve the problems, but due to time limitations this was not investigated.

Common in these simulations were the following parameters:
4.4. Simulation Results

Figure 4.11: Plot showing the amplitude of the variation in density at the interface between the two fluids. Increasing amplitude suggest stable solution. "Instable 4" has $\rho_1/\rho_2 = \frac{1}{2}$ and "Instable 5" has $\rho_1/\rho_2 = 2$.

Table 4.5: Parameters used to simulate the instable plot shown in Figure 4.12.

<table>
<thead>
<tr>
<th>Piston width $\mu$m</th>
<th>Frequency [Hz]</th>
<th>Amplitude $\mu$m</th>
<th>$\sigma$ [J/m$^2$]</th>
<th>Pressure [Pa]</th>
<th>$\eta_1$ [Pa s]</th>
<th>$\eta_2$ [Pa s]</th>
<th>$\rho_1$ [kg/m$^3$]</th>
<th>$\rho_2$ [kg/m$^3$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>640</td>
<td>250, 500, 1000</td>
<td>3</td>
<td>0.0018</td>
<td>2000</td>
<td>0.001</td>
<td>0.002</td>
<td>1000</td>
<td>900</td>
</tr>
</tbody>
</table>

4.4.1 Summary

Our investigations showed that all simulations yielded instable solutions as the amplitude in density increased along the channel. We tried to find certain parameters that would influence this effect to a greater extend, but no conclusive result was found. The simulations are simply too time consuming to investigate all parameters in such details.

As we increase the length of the channel the amplitude of the variation in density increase exponentially along the channel. This was not apparent in the first simulations, because the channel was simply too short.
Figure 4.12: Plot showing the amplitude of the variation in density at the interface between the two fluids. The frequency of the piston is the only parameter being changed. Tendency lines are exponential approximations.
Chapter 5

Conclusion

We have presented theoretical models for the Young-Laplace equation, Poiseuille flow with two fluids in a microchannel, transient times for Poiseuille flow in both a circular pipe and between two infinite parallel plates and derived a dispersion relation for small perturbations. All theoretic models have been simulated using the CFD-ACE software in order to investigate the predictability of the simulated systems.

Most of these simulations showed good agreement with theory, however, we did not achieve convincing correlation between the derived dispersion relation and the corresponding simulated model. This could either be explained by wrong usage of the simulation program or excessive simplification of the theoretic model.

Previous theoretic models presented by [3, 5] showed that it should be possible to yield instable oscillations of the interface in a fluidic system with two different fluids present. We have showed that this is indeed the case when using the CFD-ACE simulations program with Poiseuille flow profiles. We did not obtain any stable solutions to this flow profile.

It is difficult to make any precise conclusion from our limited simulation data, regarding the instability of the perturbed interface. We have showed that it is possible to achieve an exponentially growing displacement of the interface, resulting in an elongated interface length. This result is, however, not valid for all simulation parameters and should not be used as a guideline.
Bibliography


Appendix A

Maple code

A.1 Two fluids

```maple
restart:
with(plots):
with(DEtools):
E := eta[1]/eta[2];
A[1] := 1/eta[1]*(h^2*(1-E)-H^2)/(h*(1-E)-H);
simplify(%);
eta[1] := 8.4e-2:
eta[2] := 1e-3:
E := eta[1]/eta[2];
H := 100e-6:
h := H/2:
p1 := plot(h^(-2)*(eta[2]+eta[1])*1/2*v[2](Z*H),Z=0..1/2,color=red):
p2 := plot(h^(-2)*(eta[2]+eta[1])*1/2*v[1](Z*H),Z=1/2..1,color=green):
p3 := plot([0.75,10,'Water'],align=RIGHT):
p4 := plot([0.75,9,"Corn oil"],align=RIGHT):
display(p1,p2,p3,p4,p5,p6,
axes=BOXED,
axesfont = [TIMES,ROMAN,18],
font = [TIMES,ROMAN,14],
labelfont = [TIMES,ROMAN,14],
labels = ['H','GED'],
view = [0..1,0..12]);
```

A.2 Cylindrical pipe

```maple
restart:
with(plots):
```

35
A.2. CYLINDRICAL PIPE

APPENDIX A. MAPLE CODE

```maple
> J[0] := y -> BesselJ(0,y):
> J[1] := y -> BesselJ(1,y):
> lambda := k -> BesselJZeros(0,k):
> v := r ->
> sum(8/(J[1](lambda(k))*lambda(k)^3)*J[0](lambda(k)*r/a),k=1..N);
> V := R -> subs(r=R*a,v(r));

> N:=10:
> p1 := plot(
> V(R),R=0..1,
> axes = BOXED,
> axesfont = [TIMES,ROMAN,18],
> color = BLACK,
> font = [TIMES,ROMAN,14],
> labels = ['R','V'],
> labelfont = [TIMES,ROMAN,14],
> view = [0..1,0..1]);
> p2 := plot(1-R^2,R=0..1,
> style = PATCH,
> axes = BOXED,
> axesfont = [TIMES,ROMAN,18],
> color = BLACK,
> font = [TIMES,ROMAN,14],
> labels = ['R','V'],
> labelfont = [TIMES,ROMAN,14],
> view = [0..1,0..1]);
> display(p1);
> display(p2);

> p3 := seq(plot(1-R^2-V(R),R=0..1,
> color=COLOR(RGB,rand()/10^12,rand()/10^12,rand()/10^12),
> legend = convert(M = N,string)),N=1..5):
> display(p3,
> style = PATCH,
> axes = BOXED,
> axesfont = [TIMES,ROMAN,18],
> font = [TIMES,ROMAN,14],
> labels = ['R','V'],
> labelfont = [TIMES,ROMAN,14]);

> tau := k -> a^2/(nu*lambda(k)^2):
> u1 := r ->
> sum(8/(J[1](lambda(k))*lambda(k)^3)*J[0](lambda(k)*r/a)*exp(-T*tau(1)/
> tau(k)),k=1..N);
> U := R -> subs(r=R*a,u1(r));

> N := 10:
> for i from 0 to 13 do:
> T := (i/5)^2;
> p[i] := plot(U(R),R=0..1,
> color = BLACK);
> od:
```

36
A.3 Parallel plates

```maple
restart:
with(plots):
lambda := n -> sqrt(nu)/a*(Pi/2+n*Pi);
v := z -> 1/a^2*(a^2 - z^2 -
sum(32*a^2*(-1)^n*exp(-lambda(n)^2*s)*cos(lambda(n)*z/sqrt(nu))/(Pi^3*
(1+6*n+12*n^2+8*n^3)),n=0..N));
N := 10;
a := 50e-6;
nu := 1e-6;
for i from 0 to 13 do:
T := (i/8)^3;
t := lambda(0)^(-2)*T;
p[i] := plot(v(Z*a),Z=-1..1,color=BLACK):
od:
display(seq(p[i],i=0..13),
axes=BOXED,
scaling = CONSTRAINED,
axesfont = [TIMES,ROMAN,18],
font = [TIMES,ROMAN,14],
labelfont= [TIMES,ROMAN,14],
labels = ['Z','HEST'],
view = [0..1,0..1]);
N:=10;z:=0:
a:=50e-6:
nu:=1e-6:
V := s -> 1/a^2*(a^2 - z^2 -
sum(32*a^2*(-1)^n*exp(-lambda(n)^2*s)*cos(lambda(n)*z/sqrt(nu))/(Pi^3*
(1+6*n+12*n^2+8*n^3)),n=0..N));
plot(V(s*lambda(0)^(-2)),s=0..10,
axes=BOXED,
axesfont = [TIMES,ROMAN,18],
font = [TIMES,ROMAN,14],
labelfont= [TIMES,ROMAN,14],
labels = ['T','KO'],
view = [0..10,0..1.01],
color=BLACK
);
seq(evalf(lambda(n)^(-2)),n=0..5);
```