Eindhoven University of Technology

MASTER

Liquid metal flows in annular channels
COMSOL simulations of the scaling behavior of the velocity flux at the surfaces in concentric annular channels driven by electromagnetic forces

Arts, W.

Award date:
2013

Disclaimer
This document contains a student thesis (bachelor's or master's), as authored by a student at Eindhoven University of Technology. Student theses are made available in the TU/e repository upon obtaining the required degree. The grade received is not published on the document as presented in the repository. The required complexity or quality of research of student theses may vary by program, and the required minimum study period may vary in duration.

General rights
Copyright and moral rights for the publications made accessible in the public portal are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

• Users may download and print one copy of any publication from the public portal for the purpose of private study or research.
• You may not further distribute the material or use it for any profit-making activity or commercial gain

Take down policy
If you believe that this document breaches copyright please contact us providing details, and we will remove access to the work immediately and investigate your claim.
LIQUID METAL FLOWS IN ANNULAR CHANNELS

COMSOL simulations of the scaling behaviour of the velocity flux at the surface in concentric annular channels driven by electromagnetic forces

WOUTER ARTS

Science and Technology of Nuclear Fusion

Supervisors:
Dr. R.J.E Jaspers
Ir. M.A. Jakobs

Eindhoven University of Technology
Department of Applied Physics

Eindhoven, The Netherlands, July 2013
Conditions at the first wall of a nuclear fusion reactor are harsh, due to a high heat load and large neutron fluxes. Especially at the divertor. Tungsten and carbon are commonly considered as a material for the divertor. A possible alternative is to cover the outer wall of the divertor with a layer of a liquid metal (e.g. lithium). This liquid metal layer has the following advantages: no degradation of the material properties, self healing and low recycling. To drive and control such a liquid metal flow is challenging. Therefore a pilot study has been started to investigate numerically the basic effects in such a flow. The goal of the simulations is to find the influence of a top layer, driven by a system of concentric annular channels, on the the scaling of the velocity flux at the surface of a liquid metal with the control parameter, the magnetic field and the electric current. For experimental convenience the geometry is made of several concentric annular channels which are place at the bottom of a round bath. The flow is driven by the Lorentz force, which is induced by a radial electric field and a vertical magnetic field. The simulations are done in COMSOL multiphysics and have been validated using benchmarks in annular systems. Using this simulation in combination with a theoretical analysis three areas of different scaling in a single annular channel have been identified. For small magnetic field and currents the velocity flux at the surface scales linearly, increasing the magnetic field leads to magnetic braking by induced currents. If the current is increased from the linear regime sub-linear scaling is observed due to secondary flows. A top layer influences this scaling by reducing the size of the regime with linear scaling, the shape and scaling in the regimes stays the same. It has been found that large currents are needed to drive a flow in the divertor due to magnetic braking. To induce a secondary flow even larger currents are needed.
ACKNOWLEDGMENTS

This dissertation thesis finalizes my educational career which has spanned over more than two decades. There are too many people I have to thank for making this possible, but my main gratitude goes to my family, friends and teachers. Besides finishing my educational career this thesis also marks the end of my time in the fusion group at the Eindhoven University of Technology, I also want to thank everyone who made this a special time. Especially Roger Jaspers and Merlijn Jakobs for supervising my graduation project. While not supervising my project a major contribution in the guidance came from Leon Kamp, something I really appreciated. And finally, I do not know if I really want to do it, but thank you Peter for doing exactly the same things as I did for the past eight years.
CONTENTS

1 INTRODUCTION 1
  1.1 Nuclear fusion .............................. 1
  1.2 Divertor .................................. 1
  1.3 Goal ...................................... 6
  1.4 Methods .................................. 6
  1.5 Outline .................................. 7

2 MAGNETOHYDRODYNAMICS 9
  2.1 The magnetohydrodynamic equations .......... 9
  2.2 Buckingham’s Π theorem ...................... 10
  2.3 Scaling of the velocity flux at the surface .... 12

3 NUMERICAL FRAMEWORK 19
  3.1 COMSOL modules .............................. 19
  3.2 Mathematical framework ...................... 20
  3.3 Meshing .................................... 22

4 BENCHMARKING 23
  4.1 Flow between two cylinders duct .............. 24
  4.2 Straight square duct flow .................... 27
  4.3 Annular duct with square cross section ....... 29

5 MAGNETOHYDRODYNAMIC SCALING BEHAVIOUR 33
  5.1 Individual annular channels channels .......... 34
  5.2 Geometry with a top layer .................... 47

6 CONCLUSION, DISCUSSION AND OUTLOOK 53
  6.1 Conclusion ................................ 53
  6.2 Discussion ................................ 55
  6.3 Future directions ........................... 57

A BUCKINGHAM’S Π THEOREM 61

BIBLIOGRAPHY 65
# List of Variables

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>(\vec{E})</td>
<td>Electric field</td>
</tr>
<tr>
<td>(\rho_E)</td>
<td>Electric charge density</td>
</tr>
<tr>
<td>(\varepsilon_0)</td>
<td>Electric permittivity of vacuum</td>
</tr>
<tr>
<td>(\vec{B})</td>
<td>Magnetic field</td>
</tr>
<tr>
<td>(\mu_0)</td>
<td>Magnetic permeability of vacuum</td>
</tr>
<tr>
<td>(\vec{j})</td>
<td>Current density</td>
</tr>
<tr>
<td>(\rho)</td>
<td>Density</td>
</tr>
<tr>
<td>(\vec{v})</td>
<td>Velocity</td>
</tr>
<tr>
<td>(\mu)</td>
<td>Dynamic viscosity</td>
</tr>
<tr>
<td>(\vec{f})</td>
<td>Force</td>
</tr>
<tr>
<td>(q)</td>
<td>Electrical charge</td>
</tr>
<tr>
<td>(\sigma)</td>
<td>Electrical conductivity</td>
</tr>
<tr>
<td>(Q)</td>
<td>Velocity flux</td>
</tr>
<tr>
<td>(a)</td>
<td>Width of the channels</td>
</tr>
<tr>
<td>(\Pi_1)</td>
<td>Dimensionless Velocity flux at the surface</td>
</tr>
<tr>
<td>(\Pi_2)</td>
<td>Dimensionless velocity</td>
</tr>
<tr>
<td>(\Pi_3)</td>
<td>Dimensionless density</td>
</tr>
<tr>
<td>(\Pi_4)</td>
<td>Dimensionless current</td>
</tr>
<tr>
<td>(\Pi_5)</td>
<td>Dimensionless magnetic field</td>
</tr>
<tr>
<td>(\text{Ha})</td>
<td>Hartmann number</td>
</tr>
<tr>
<td>(I_D)</td>
<td>Dimensionless Current</td>
</tr>
</tbody>
</table>
INTRODUCTION

1.1 NUCLEAR FUSION

Nuclear fusion is a nearly inexhaustible energy source that does not produce any greenhouse gasses. In contrast to nuclear fission it fuses two light nuclei together rather than splitting a heavy nucleus.

To make two nuclei fuse a high energy is used. In many currently proposed nuclear fusion reactor deuterium and tritium particles are typically fused at a temperature of 150 million degrees centigrade to a helium particle and a neutron. At this temperature the gas is fully ionized into a plasma.

The most promising design for a nuclear fusion reactor at the moment is a tokamak. In a tokamak magnetic fields, in the shape of a torus, are used to confine the plasma. To prove that the concept of a tokamak works ITER is being build. ITER is a tokamak in which the fusion reactions produces more energy than is necessary to sustain the reaction.

At the edge of the plasma in a tokamak the so-called scrape off layer is located, which touches the wall in the part of the reactor called the divertor. A sketch of this system is given in figure 1.

1.2 DIVERTOR

In the fusion reaction 80% of the energy is released into the neutrons and 20% into the helium ions. The energy of the helium ions is released through the scrape off layer to the divertor. The area to which this heat energy is released is relatively small (typically few m²) compared to the volume in which the energy is produced (typically many 100 m³). This leads to a large heat flux at the divertor. The estimated steady-state heat flux for ITER is in the order of 10 MW/m² \[13\]. To be able to withstand these conditions the divertor should be made of a material that is a good heat conductor with a high melting temperature.

Besides the steady-state behaviour, the heat output also has a transient behaviour. So called edge located modes (ELMs) occur multiple times per seconde due to plasma instabilities. For very brief time periods the heat flux can reach values above 100 MW/m² \[13\]. Therefore
the divertor material should have a good thermal shock resistance.

Next to the large heat loads the divertor is also subject to large neutron and ion fluxes that constantly bombard the divertor. The neutrons cause structural changes in the material due to collisions with lattice atoms. This results in swelling of the materials, dislocations of the lattice atoms and activation of the material, thereby changing the properties of the material to undesired properties.

Besides the physical changes that happen to the material due to the bombardment of neutrons, the ions interact with the material, causing swelling and chemical reactions. The plasma exists for a large part out of hydrogen isotopes (deuterium and tritium) that are highly reactive radicals. In a nuclear fusion reactor only a small amount of tritium is allowed and because it is radioactive and expensive the divertor material should have a low reactivity with tritium.
This can be summarized in the following properties that a divertor material should have [17]:

- High heat conductivity and capacity
- High melting point
- Large thermal shock resistance
- Little degradation of the material properties
- Low activation
- Low tritium retention
- Low erosion yield
- Low plasma pollution of heavy ions

No material has yet been found which has all of these properties. The materials that are currently be considered to be used in ITER are tungsten and carbon fiber composites (CFC). These materials are not ideal to use. CFC has the problem that it reacts easily with tritium. Tungsten has the problems that the loss of material into the plasma can cause a disruption and that it is vulnerable to structural changes due to neutron radiation and heat cycling.

A proposed mechanism to overcome some of these problems is to cover the divertor surface with a liquid metal. The advantage of a liquid layer is that they have some properties that a solid material does not have, these are[18]:

- No degradation of the material properties
- The material is self healing
- Low recycling

The material does not degraded because the material constantly refreshes itself with new material. The material is self healing because if for whatever reason (ELM, hotspot) the material would evaporate, new material can be brought to the damaged area. If a plasma particle hits the liquid layer it has a high chance to be absorbed into the liquid, this high absorption chance is called low recycling. This principle reduces the plasma pollution as impurities are removed out of the reactor, increasing the plasma performance.

Proposed liquids have been gallium, tin and lithium. The advantages of tin and gallium over lithium are that they can operate at higher temperatures. The advantages of lithium are: the low atomic mass, it lowers the impurities due to a low recycling rate in the
plasma and it could potentially increase the plasma performance.

The interaction of the liquid metal and the fusion plasma is a major challenge in designing a liquid metal divertor. The flow of metals inside a tokamak induces currents within the liquid metal resulting in a perturbation of the magnetic topology inside the reactor, potentially reducing the confinement of the particles. Therefore the flow of these metals and the interaction must be well understood.

Concepts of liquid metal divertors has been studied my multiple universities. Full liquid wall concepts have been proposed for example by the University of Los Angeles [6]. In Princeton they have even tested liquid lithium inside a tokamak [8]. None of the published concepts had a system in which they could locally control the liquid metal flow in the divertor.

A locally controlled liquid metal flow in a divertor would allow for active control of the divertor performance during its operation. Undesired properties of the liquid metal in the divertor could be prevented. An example of this is, if the liquid layer becomes too thin at a certain position extra material from different locations could be moved to thicken the layer by actively moving that material. Another example is to mitigate hotspots, by increasing the flow rate in the location of a hotspot to lower the temperature of the divertor at that location.

The part of the flow which must be controlled in the divertor is the free surface flow that is exposed to the plasma. The flow can be controlled by using the strong magnetic fields of a tokamak, in combination with a current. To keep the plasma facing part fully liquid this current could be applied on channels below the surface which drive the layer above. In such a system the liquid metal flow can be controlled by locally controlling the applied current in the divertor.

The concept of such a locally controllable liquid metal flow in a divertor has not yet been published. To be able to control such a flow the influence of the top layer on the total flow must be known. Therefore the magnetohydrodynamic behaviour of a liquid metal flow in a system with multiple channels joined on top by a liquid layer must be studied.

Besides the concepts using flows of liquid metals in the divertor alternative uses of liquid metals in a divertor have been proposed. A promising example is the capillary pore system (CPS) [7]. CPS uses a porous structure which sucks up a liquid metal, covering the entire structure. The main advantages of CPS is that induced currents and magnetic fields are less of a problem, as the velocity of the liquid
metal is very low. The main disadvantage of CPS over a liquid metal flow is that the heat cannot be convected out of the divertor. The remainder of this report will focus on the concept of using liquid metal flows in divertors instead of other concepts like CPS.

The study that will be performed is more general applicable than just on fusion. The magnetic fields and currents will be in the range that is achievable with common magnetic fields and moderate currents. In this way the results are also useful for other purposes of liquid metals whilst still being relevant for liquid metal divertors.

For both experimental and theoretical reasons an axial symmetric geometry is a convenient geometry to study a liquid metal flow, without losing it generality. The experimental reason for this is that experiments without begin and end losses can be done, and that no extra pumps or transport systems are needed to move the liquid. From a theoretical view this is interesting because effects of curved channels can be studied. The symmetry is theoretically also convenient because the equations can be solved in cylindrical coordinates, which leads to easier equations than most other geometries, except straight channels.

The hydrodynamic behaviour of liquid metals has already been widely studied. This field of research was started by Hartmann in 1937 in straight ducts [12]. Analytical solutions have been found for the velocity fields in multiple geometries (e.g. flow between parallel plates, strait ducts, etc.). During the years this research has been expanded towards other geometries. For experimental reasons an annular set-up is often studied [2]. Most of this research has been focussed on closed channels with either radial magnetic fields or an axial magnetic field [14]. The liquid was forced by applying, perpendicular to the magnetic field, an electric field.

No study has been published in a geometry that would be relevant for the concept of a locally controlled liquid metal divertor. Such a geometry would consist of multiple electromagnetically driven chan-

![Figure 2: Geometry of a set-up with multiple concentric annular channels.](image)

On the bottom of the container the channels are drawn. Over these channels a potential difference can be applied. In combination with a magnetic field this current exerts a force on the liquid inside the container.
nels, which drive a bulk layer on top. As explained it is most conve-
nient to do this in an annular set-up. This geometry is depicted in
figure 2, in here the magnetic field is in vertical direction, the current
in radial direction and the resulting flow in azimuthal direction.

1.3 GOAL

To be able to design a locally controllable liquid metal divertor the
amount of fluid that is be transported by the divertor using the con-
trol parameters (magnetic field, electric current) must be known. This
dependence of amount of fluid that is driven by a magnetic field and
current will be studied in this report. To be able to compare the re-
sults of both an experiment and a theoretical analysis the velocity flux
at the surface will be used as a reference parameter. To use a liquid
metal flow in a divertor the influence of the top layer on response of
the flow to the control parameters must be know, especially in what
way the velocity flux at the surface scales with these parameters. To
study this the following research question has been formulated:

What is the influence of a top layer, driven by a system of concentric annu-
lar channels, on the the scaling of the velocity flux at the surface of a liquid
metal with the control parameter, the magnetic field and the electric current.

To be able to compare theoretical predictions with experimental re-
results a secondary question must be answered:

How does the scaling of the velocity flux at the surface change if the mag-
netic field is is not perfectly parallel and perpendicular to the surface

To answer this question a simulation will be made with two dif-
ferent magnetic configurations which can simulate: a magnetohydro-
dynamic flow in a system of annular channels for different magnetic
field and currents in the relevant domain. To limit the scope of both
the research question and the goal the relevant domain will be lami-
nar flows, because in this range the relevant scaling behaviour is ex-
pected to happen. The forcing in the simulations is due to the Lorentz
force from a magnetic field that has mainly components in vertical
direction and an applied potential over the channels. From this simu-
lation the velocity flux at the surface will be obtained.

1.4 METHODS

The work will be mainly be based on numerical simulations. To sup-
port the simulations the scaling behaviour is also analyzed from a
theoretical point of view. To be able to compare the simulations to
experimental data a pilot experiment has been designed. The set-up
Figure 3: Set-up of annular channels joined on top by a bath. This geometry is used in a pilot experiment to study the behaviour of a liquid metal flow, the same dimensions are used in the simulations.

and experiments are described in the report of Koelman [15]. But the experimental data does not yet offer conclusive data to underpin the simulations. Many of geometrical choices in the simulation have been based on this experiment, like the channel width, depth and inner radius, position and strength of the bar magnets, etc. The geometry of the experiment is depicted in figure 3. The material that was used in the experiment was galinstan, therefore its properties are used in the simulation.

The simulation software that is used is COMSOL version 4.3a. COMSOL is a widely used simulation package that can solve differential equation using a finite element method. Dedicated MHD-codes also exist, but due to the availability and adaptability of COMSOL (it easily be extended to solve combined physical phenomena) it was chosen to develop a simulation of a liquid metal flow in COMSOL.

To check whether the simulation is valid and correct in the range where scaling is studied the simulations are benchmarked against known solutions and compared to other numerical simulations. Using the benchmarked simulation, studies of the relevant geometries will be made. From these simulations the scaling behaviour at the surface of the liquid metal flow is obtained.

1.5 OUTLINE

The theoretical analysis is explained in chapter 2 by reviewing the magnetohydrodynamic theory. From this theory simple qualitative scaling laws are derived. The numerical framework that is used in COMSOL and its relation to the theory is described in chapter 3. The benchmarks are explained and done in chapter 4. In chapter 5 the
results from the simulation will be explained and analyzed to give quantitative values for the scaling behaviour. From this work the conclusion, discussion and future directions will be given.
In this chapter a theoretical analysis of liquid metal flows is made. First the magnetohydrodynamic (MHD) equations are presented, along with some other relevant equations. Buckingham’s Π theorem is used to find the relevant parameters of the equations. After that the scaling of the velocity flux at the surface with these parameter is studied.

2.1 THE MAGNETOHYDRODYNAMIC EQUATIONS

The MHD theory describes the motion of a conducting fluid and its interaction with magnetic and electric fields. The MHD equations consist of Maxwell’s laws for electromagnetism, the Navier-Stokes equation for hydrodynamics and the conservation of mass. The equations are:

Maxwell’s equations:
\[ \nabla \cdot \vec{E} = \frac{\rho E}{\varepsilon_0} \]  
(1)
\[ \nabla \cdot \vec{B} = 0 \]  
(2)
\[ \nabla \times \vec{E} = -\frac{\partial \vec{B}}{\partial t} \]  
(3)
\[ \nabla \times \vec{B} = \mu_0 \vec{J} + \mu_0 \varepsilon_0 \frac{\partial \vec{E}}{\partial t} \]  
(4)

Navier-Stokes equation:
\[ \rho \left( \frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v} \right) = -\nabla p + \mu \nabla^2 \vec{v} + \rho E \vec{E} + \vec{J} \times \vec{B} + \vec{F}_{\text{other}} \]  
(5)

Conservation of mass:
\[ \frac{d\rho}{dt} + \rho \nabla \cdot \vec{v} = 0 \]  
(6)

The definition of the terms are:

- \( \vec{E} \): Electric field
- \( \rho E \): Electric charge density
- \( \varepsilon_0 \): Electric permittivity of vacuum
- \( \vec{B} \): Magnetic field
- \( \mu_0 \): Magnetic permeability of vacuum
- \( \vec{J} \): Current
- \( \rho \): Density
- \( \vec{v} \): Velocity
- \( \mu \): Dynamic viscosity
Maxwell’s equations in combination with the Navier-Stokes equation and the conservation of mass describe: the electric field, the magnetic fields and the flow of a liquid metal. Two other useful electromagnetic relations are the Lorentz force and Ohm’s law. The Lorentz force gives the relation between the electromagnetic fields and the force exerted on a charged body (this equation was already used in 5):

\[ \vec{F} = \rho \vec{E} + \vec{J} \times \vec{B} \]  

(7)

The current is induced by two mechanisms. The first one is the applied electric field the second mechanism is the motion through an applied magnetic field of the fluid. This combination leads to Ohm’s Law:

\[ \vec{J} = \sigma \left( \vec{E} + \vec{v} \times \vec{B} \right) \]  

(8)

Solving the MHD equations for the appropriate boundary conditions yields the electric field, the magnetic fields and the velocity field of the flow. These differential equations have in general no analytical solution, only in some special cases. Therefore numerical methods are needed to solve these equations. The numerical framework that is used to solve these equations in the simulations is described in chapter 3.

In these equations no induced magnetic fields have been assumed. This assumption is valid because the magnetic Reynolds number is small for typical conditions encountered in our applications. The magnetic Reynolds number is a dimensionless number that gives an estimate of the effects of the magnetic advection to the magnetic diffusion \( (R_m = \mu_0 \sigma L v_0) \). When this number is small it can be assumed that the induced magnetic field are small enough that they can be neglected.

2.2 Buckingham’s Π theorem

As can be seen in the Navier-Stokes equation 5 the flow responds to the forcing of the flow. This forcing can either be internal or external. In the simulations an internal forcing by means of an applied current and magnetic field is used. To study the scaling behaviour of the flow on the forcing Buckingham’s Π theorem is used [3]. Buckingham’s Π theorem allows to create a complete set of independent dimensionless variables of a problem.

In the simulations the velocity flux at the surface is the output parameter of interest. There are three reasons for that. The first one is that if the velocity flux at the surface is important in the concept of liquid metal divertors in tokamaks. The velocity flux at the surfaces determines the amount of heat the flow can absorb at the surface. The
second reason is for experimental convenience, in an experiment it is relatively easy to determine the velocity flux at the surface, opposed to determining the total velocity flux rate. The third reason is that one expects the velocity flux to change with a changing forcing, as the velocity is expected to change.

The derivation of the results in Buckingham’s \( \Pi \) theorem can be found in the appendix. The results are presented in this section. The velocity flux at the surface is a function of:

\[
Q_s = f (v, \rho, \mu, a, I, B, \sigma, \mu_m)
\]

The definitions of the new terms are:

- \( Q \): Velocity flux at the surface
- \( I \): Applied current on the channels
- \( \sigma \): Electrical conductivity
- \( a \): Width of the channels

There are nine variables and four physical quantities (length, time, mass and current). According to Buckingham’s \( \Pi \) theorem there are \((9 - 4) = 5\) independent dimensionless numbers. These numbers can be obtained from this function:

\[
\Pi = Q_s^a v^a_2 \rho^a_3 \mu^a_4 a^a_5 I^a_6 B^a_7 \sigma^a_8 \mu^a_9
\]

As repeating variables \( \mu, a, \sigma, \mu_m \) are chosen. These are chosen because they allow to compare the different geometries. This results in the following dimensionless numbers:

\[
\begin{align*}
\Pi_1 &= \frac{Q_s \sigma \mu m \mu_m}{a} \quad \text{Dimensionless velocity flux at the surface} \quad (11) \\
\Pi_2 &= \frac{v a \sigma \mu_m}{\mu} \quad \text{Dimensionless velocity} \quad (12) \\
\Pi_3 &= \frac{\rho}{\mu \sigma \mu_m} \quad \text{Dimensionless density} \quad (13) \\
\Pi_4 &= \frac{I m \mu_m \sigma}{\mu} \quad \text{Dimensionless current} \quad (14) \\
\Pi_5 &= B a \sqrt{\frac{\sigma}{\mu}} \quad \text{Dimensionless magnetic field} \quad (15)
\end{align*}
\]

The last dimensionless number \((\Pi_5)\) is in the literature known as the Hartmann number \((H_a)\).

As stated before in the simulations the parameter of interest is the velocity flux at the surface. The dimensionless variant of this is the ‘dimensionless velocity flux’ \( \Pi_1 \). The typical velocity \( \Pi_2 \) could also have been used as a output parameter of the system, but this will not be done. The reason is that the velocity flux at the surface is more interesting to study for experimental and divertor reasons. The dimensionless number \( \Pi_3 \) is a parameter of the system and can not be
changed. The dimensionless current $\Pi_4$ is one of the variables that can be changed. It can be changed by the applied voltage over the channels. A larger value of $\Pi_4$ will result in a larger forcing of the fluid. Similarly the Hartmann number $\Pi_5$ influences the flow.

From Buckingham’s $\Pi$ theorem it can be concluded that the dimensionless current $\Pi_4$ and the Hartmann number $Ha$ influence the flow, which can be characterized by both the dimensionless velocity flux at the surface $\Pi_1$ and dimensionless velocity $\Pi_2$. For the simulation the scaling behaviour of the velocity flux at the surface on both the current and magnetic field will be studied. For better readability the dimensionless current $\Pi_4$ will be assigned the variable $I_D$.

2.3 Scaling of the velocity flux at the surface

To study the scaling behaviour of such a flow it would be very convenient to have an analytic solution. But no analytic solution is known for open annular channels. Solutions do exist for straight channel flows. Since that these analytic solutions offer a very good insight in the scaling behaviour they will be presented here first. After that the changes for an open annular channels will be given.

2.3.1 Scaling in straight ducts

Two solutions exist for straight channel flows, for a closed duct and for a flow between parallel plates. Both solutions offer basically the same information on scaling. The result for a closed duct is explained in this chapter since that is the most similar flow to an flow in an open annular channel.

![Figure 4: A straight duct flow. The magnetic field is in the z-direction, the flow is in the x-direction.](image-url)
A flow of a liquid metal under electromagnetic forcing is called a
Hartmann flow. The closed duct flow is analyzed to be able to un-
derstand the physical phenomena that happen in a Hartmann flow. A
sketch of the system is given in figure 4. The flow is driven by a con-
stant pressure gradient. Such a gradient can be achieved by applying
an electric field in the y-direction. This yields a constant current in
the same direction. The solution found by Shercliff [16] of the flow is:

\[
v = \sum_{i=1,3,5}^{\infty} v_i(y) \cos(\lambda_i x) \tag{16}
\]

with:

\[
\lambda_i = \frac{i\pi}{2a},
\]

\[
k_i = 2 \frac{\sin(\lambda_i a)}{\lambda_i a},
\]

\[
p_{i_{1,2}} = \frac{1}{2} \left( \text{Ha} \mp \sqrt{\text{Ha}^2 + 4\lambda_i^2} \right)
\]

\[
\alpha_{i_{1,2}} = \sinh(p_{i_{1,2}})
\]

\[
f_i(y) = \alpha_{i_{1}} \cosh(p_{i_{1}} y) - \alpha_{i_{2}} \cosh(p_{i_{2}} y)
\]

\[
v_i(y) = \frac{k_i}{\lambda_i^2} \left( 1 - \frac{f_i(y)}{f_i(1)} \right)
\]

In figure 5 the profile of the flow is plotted for several different Hart-
mann numbers.

As can be seen in figure 5 the velocity profile at higher Hartmann
number tend to level out. The profile gets flatter because when a
conductor moves through a magnetic field a current is induced. This

![Figure 5: Flow profiles for different Hartmann numbers in a straight duct, the lines for Ha = 0 and Ha = 1 barely differ.](image-url)
current is proportional to the velocity. The induced current exerts a
lorentz force on the fluid in opposite direction to the motion, thereby
leveling the velocity profile. Another effect from the induced current
is that the maximum velocity has an inverse relation with the magnetic
field.

The maximum flow velocity changes under a changing magnetic
field because the magnetic field applies a forcing on the liquid metal
in two different ways. These forces are in opposite direction. The first
force is due to the applied current and the applied magnetic field. The
other force is due to the induced current and the magnetic field. The
induced current is proportional to the velocity and the magnetic
field. This causes a non-linear relation for the magnetic forces:

\[ F_{\text{mag}} \propto J_{\text{applied}} \times B + J_{\text{induced}} \times B \]

\[ \propto J_{\text{applied}} \times B + (\sigma v \times B) \times B \]

This force leads to a non-linear scaling behaviour of the scaling of
the velocity flux. The exact solution for the velocity flux is:

\[ Q = \sum_{i=1,3,5}^{\infty} \frac{k_i^2}{\lambda_i^2 \left(1 - F_i(1) \frac{f_1}{f_2} \right)} \]

With: \( F(1) = \int_0^1 f(y) \, dy \)

The volumetric rate is plotted in figure 6 versus the Hartmann num-
ber for a constant pressure gradient. The pressure gradient scales lin-
early with the product of the current and the magnetic field. From this
graph the scaling behaviour for linear channels can be understood.
As can be seen from the graph the volumetric scales linearly with the Hartmann number until a certain critical value near one. After this threshold the velocity flux drops for higher Hartmann number. The velocity flux at the surface is expected to scale in very similar way to the volumetric rate in a channel.

From graph 6 it can also be concluded that the velocity flux scales linearly with the current. This is because the pressure gradient scales linearly with the product of the magnetic field and current and the fact that this graph is valid for all pressure gradients.

2.3.2 Scaling in an open annular channels

The free surface at top of the flow leads to a different boundary condition for the flow. The top boundary now has a stress free boundary condition instead of a no-slip condition. Therefore the flow can have a velocity at that boundary.

When a current is applied to an annular channel the current density is not constant anymore, since the current goes through a larger area on the outside of the channel than on the inside. This yields a $1/r$ dependence of the current density.

Due to the fact that an annular channel is curved the flow will behave differently. What changes happen and why will be explained by looking at the Navier-Stokes equation in cylinder coordinates.

$$r:$$
$$\rho \left( \frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + \frac{u_r}{r} \frac{\partial u_r}{\partial \phi} + u_z \frac{\partial u_r}{\partial z} - \frac{u_\phi^2}{r} \right) = \frac{\partial p}{\partial r} +$$
$$\mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_r}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 u_r}{\partial \phi^2} + \frac{1}{r^2} \frac{\partial^2 u_r}{\partial z^2} - \frac{u_r}{r^2} - \frac{2}{r} \frac{\partial u_\phi}{\partial \phi} \right) + F_r$$

$$\phi:$$
$$\rho \left( \frac{\partial u_\phi}{\partial t} + u_r \frac{\partial u_\phi}{\partial r} + \frac{u_r}{r} \frac{\partial u_\phi}{\partial \phi} + u_z \frac{\partial u_\phi}{\partial z} + \frac{u_\phi u_r}{r} \right) = -\frac{1}{r} \frac{\partial p}{\partial \phi} +$$
$$\mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_\phi}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 u_\phi}{\partial \phi^2} + \frac{1}{r^2} \frac{\partial^2 u_\phi}{\partial z^2} - \frac{u_\phi}{r^2} - \frac{2}{r} \frac{\partial u_r}{\partial \phi} \right) + F_\phi$$

$$z:$$
$$\rho \left( \frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + \frac{u_r}{r} \frac{\partial u_z}{\partial \phi} + u_z \frac{\partial u_z}{\partial z} \right) = \frac{\partial p}{\partial z} +$$
$$\mu \left( \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial u_z}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 u_z}{\partial \phi^2} + \frac{1}{r^2} \frac{\partial^2 u_z}{\partial z^2} \right) + F_z$$
In the equations a couple of extra terms become apparent, all of these terms have either a $1/r$ or $1/r^2$ dependency. The coupling of these terms with the radius of the channel induces a secondary flow.

The secondary flow moves outwards on the top of the flow and then moves downward along the outer wall to move inwards over the bottom. The most popular explanation was given by Einstein in 1926 [5]. He used it to explain the erosion of rivers banks and the movement of tea leafs in a cup whilst stirring it.

The secondary motion is induced by the centripetal force needed by the fluid to follow the curved annular channel. A pressure gradient yields this force. The flow has a smaller velocity near the bottom and the sides of the channel. Therefore the centrifugal force is smaller there. The pressure gradient however remains intact. The pressure gradient at the bottom is thus larger than the centrifugal force inducing an inward motion of the liquid at the bottom. This effect creates a circular secondary flow in annular channel.

A part of the energy of the primary flow goes to the secondary flow. This causes a less efficient forcing of the system then a straight channel, i.e. more force is needed to keep the flow moving with the same velocity. The secondary flow becomes important when the velocity is high enough. This means that at high forcing (i.e. large $J \times B$) there is less efficient forcing than at low forcing. This leads to sub-linear scaling at large values of $J \times B$.

The magnetic field influences the flow in much the same way in annular channels as it does in straight ducts. Currents are induced by the magnetic field. These currents balance the applied currents in the center of the flow, thereby leveling the velocity profile. The velocity profile is not leveled to a flat pattern as in a straight channel but to a $1/r$ profile. This is because of the shape of the current density.

The phenomena explained in this section lead to a poorer efficiency of the forcing of the velocity flux at the surface. Therefore the scaling is influenced by this.

From these two different types of scaling it is expected that there are three regimes of different scaling of the velocity flux at the surface. The first regime is for small dimensionless currents and Hartmann numbers. In this regime linear scaling behaviour is expected, since the flow reacts linear to the forcing in this regime. This regime will be called the linear regime.
If the Hartmann number is increased at low dimensionless currents the flow velocity will reduce due to magnetic braking. This braking is induced by induced currents in the flow due to Faraday’s Law. This regime will be called the magnetic braking regime, and will occur when the Hartmann number becomes large, this effect is independent on the flow velocity.

If the dimensionless current is increased from a small Hartmann number a secondary flow is induced. This secondary flow increases for larger forcing. This leads to less efficient forcing and therefore sub-linear scaling. At a certain forcing this regime will be reached, thus for a certain $J \times B$. This regime will be called the curvature dominated regime or secondary flow regime.

In figure 7 the locations of different scaling have been plotted with the approximate boundary between these regimes. The simulations will quantify the exact shape of the boundary line.

2.3.3 Effects of a inhomogeneous the magnetic field

In an experimental set-up the magnetic field will not be exactly be homogeneous. The result of this inhomogeneity of the magnetic field will lead to a difference in the scaling behaviour of the magnetic field.

In an experimental setup, bar magnets below the container, can be used to drive the flow through the channel. This system is used in
the experiments and is depicted in figure 3. In such a set-up the magnetic field is strongest at the bottom of the channel. This is where the velocity gradient is the highest. Because of the higher magnetic field the gradient will be reduced, resulting in a lower maximum velocity at the surface of the flow.

The secondary flows will also be influenced by the inhomogeneous magnetic fields, as they move through a constantly changing magnetic field. This will result in a different velocity profile and might influence the shape and location of the boundary between the regimes of different scaling.
The MHD equations that were presented in the last chapter will be solved numerically, this will be done using a program called COMSOL multiphysics. COMSOL is a simulation package that solves differential equations using a finite element method.

This chapter starts with an introduction of the parts (modules) that are used in COMSOL. This is followed by the implementation of the magnetohydrodynamic equations in COMSOL. After that the boundary conditions for MHD-flows are explained. In the last section the phenomena with the smallest spatial dimension will be described and quantified. From this the minimum size of the mesh elements will be determined to resolve these phenomena.

3.1 COMSOL Modules

COMSOL uses modules to simulate different physical principles or mathematics. These modules can be combined to simulate multiple physical phenomena. To simulate a Hartmann flow the following modules are used:

- Laminar flow (spf)
- Poisson’s equation (poeq)
- Magnetic fields (mf)

The ‘Laminar flow’ module is used to solve the Navier-Stokes equation. This module allows to simulate laminar flow.

The ‘Poisson’s equation’ module calculates the electric fields and currents. Both the applied and induced fields are calculated using this module.

The magnetic fields are calculated using the ‘magnetic fields’ module. The magnetic field can be decoupled from the currents since the magnetic Reynolds number is small, see section 2.1. Therefore no induced magnetic field needs to be calculated.

Two different magnetic fields are used in the simulation. The first is a constant vertical magnetic field. This magnetic field is not simulated, the strength is directly used in the calculation of the Lorentz force in the Navier-Stokes equation and Poisson’s equation. The second
magnetic field is a more realistic field. This field has both vertical and radial components. This magnetic field is calculated using the magnetic fields module in COMSOL.

3.2 Mathematical Framework

3.2.1 Poisson’s equation

The electric potential is calculated using Poisson’s equation with the appropriate boundary conditions and sources terms. From Poisson’s equation the electric current is obtained. In electrostatics Poisson’s equation is defined as:

\[ \nabla^2 V = -\frac{\rho}{\varepsilon_0} \]  \hspace{1cm} (23)

Poisson’s equation has the following definition in COMSOL:

\[ \nabla \cdot (-\varepsilon \nabla V) = f \]  \hspace{1cm} (24)

This can be rewritten from Ohm’s law in the stationary state to:

\[ \nabla^2 V = \nabla \cdot (\nu \times B) \]  \hspace{1cm} (25)

A special note on this must be taken in COMSOL for cases with cylindrical coordinates. Which is relevant for doing simulations with an axial symmetry. Laplace’s operator in COMSOL is defined as:

\[ \nabla^2_{\text{COMSOL}} V = \frac{\partial^2 V}{\partial r^2} + \frac{\partial^2 V}{\partial \phi^2} + \frac{\partial^2 V}{\partial z^2} \]  \hspace{1cm} (26)

Whereas the Laplacian in cylinder coordinates is defined as:

\[ \nabla^2 V = \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial V}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 V}{\partial \phi^2} + \frac{\partial^2 V}{\partial z^2} \]  \hspace{1cm} (27)

\[ = \frac{1}{r} \frac{\partial V}{\partial r} + \frac{\partial^2 V}{\partial r^2} + \frac{1}{r^2} \frac{\partial^2 V}{\partial \phi^2} + \frac{\partial^2 V}{\partial z^2} \]

Therefore the following correction factor is need to translate from COMSOL’s Poisson’s equation to the normal one:

\[ \nabla^2 V = \nabla^2_{\text{COMSOL}} V + \frac{1}{r} \frac{\partial V}{\partial r} \]  \hspace{1cm} (28)

This system can be solved for the appropriate boundary conditions.

3.2.2 Boundary conditions for Poisson’s equation

To drive a current, a potential is applied over the channel. This can easily be applied as a boundary condition of Poisson’s equation. The other boundaries of the system are electric insulators. This means that
no current flow through the wall, i.e. $J \cdot \hat{n} = 0$. Using Ohm’s law the insulating boundary conditions can be obtained:

$$ J \cdot \hat{n} = \sigma (E + v \times B) \cdot \hat{n} = 0 \quad (29) $$

If a purely vertical magnetic field is used $B = B_0 \hat{e}_z$ the $v \times B$ term drops out since through either $(v \times B)_z = v_x B_y - v_y B_x = 0$ or $(v \times B)_z = v_r B_\phi - v_\phi B_r = 0$, thus the boundary condition becomes:

$$ E \cdot \hat{n} = 0 \quad (30) $$

This translates in a boundary condition for the potential as:

$$ \frac{\partial V}{\partial z} = 0 \quad (31) $$

If more realistic magnetic fields are used, that is a vertical field with radial components. The boundary conditions become in either cartesian or cylinder coordinates for the free surface:

**Cartesian:**

$$ \frac{\partial V}{\partial z} = v_x B_y - v_y B_x \quad (32) $$

**Cylinder:**

$$ \frac{\partial V}{\partial z} = v_r B_\phi - v_\phi B_r \quad (33) $$

The boundary conditions in combination with equation 25 given the electric potential in the channels.

### 3.2.3 Current calculation

The potential is coupled to the Navier-Stokes module through the Lorentz force on the fluid. The Lorentz force 7 uses the current and a magnetic field. The current is calculated from the potential using Ohm’s law 8:

$$ J = \sigma (E + v \times B) \quad (34) $$

$$ = \sigma (-\nabla V + v \times B) \quad (35) $$

From this current the Lorentz force can be calculated:

$$ F = J \times B \quad (36) $$

The Lorentz force is used as a volume force in the laminar flow module of COMSOL. The laminar flow module and Poisson’s equations module can together be solved for the appropriate boundary conditions and magnetic field to obtain the flow a liquid metal in COMSOL.
The mesh size determines the precision to which a solution can be found. The mesh size must be optimized to allow for fast calculations, a coarser mesh is faster calculated than a finer mesh. But a coarse mesh might not always be able to resolve all physical phenomena.

The mesh should be the finest in the places with the highest gradients, which is at the walls of the system. At the walls the Hartmann layer and side layer are located. The Hartmann layer is located at bottom of a channel and its size is given by:

$$\delta_H = \sqrt{\frac{\mu}{B^2 \sigma}}$$  \hspace{1cm} (37)

The side layers are located at the vertical walls, where electrodes are that drive the flow, their thickness is given by:

$$\delta_S = \sqrt{\frac{L}{B} \sqrt{\frac{\mu}{\sigma}}}$$  \hspace{1cm} (38)

Inside these layers multiple points are needed to correctly calculate the flow. In the simulations typically at least 10 points were used in the Hartmann layer and side layer. In the core of the flow larger element sizes are allowed. Other properties of the flow or solver can make smaller element sizes necessary, but in general these numbers yield a good approximation for the minimal element size.
BENCHMARKING

In the last chapter a numerical framework has been introduced to calculate a Hartmann flow. In this chapter the framework will be verified. In total three different benchmarks will be used to validate all parts of the simulation.

The first benchmark is a flow between to infinitely long cylinders. The fluid will be driven by a potential difference between the cylinders. By comparing the analytical solution of this flow with the results from the simulation it can be checked whether the simulation is correctly implemented in cylindrical coordinates.

The second benchmark is the flow in an infinitely long straight duct. The analytical solution of this flow will be compared to a simulation of a flow in an annular channel with a small aspect ratio. Both solutions are expected to have a similar solution because of the small aspect ratio of the annular channel. With this simulation the correct implementation of the boundary conditions and the response to the magnetic field can be validated.

The last benchmark is also a flow in an annular channel, but now with a large aspect ratio. This large aspect ratio allows for a validation of the effect of the curvature i.e. the secondary flow. This benchmark will be done by comparing the solution of the simulation with a simulation published by Vantieghem and Knaepen.

Therefore all relevant phenomena of a laminar MHD-flow in an annular channel with a forcing by a radial current and a vertical magnetic field can be validated using these benchmarks.

The comparisons will be made in the relevant range for the simulations. This range is for the Hartmann number from zero (hydrodynamic flow) to a hundred (strong magnetic braking). This covers the range in which changes are expected to happen. The electric potential is chosen at a value that the flow is laminar. The current in the comparison with the model of Vantieghem and Knaepen will be chosen to match their Reynolds numbers.
4.1 Flow Between Two Cylinders Duct

To validate the correct implementation of the system in cylindrical coordinates in COMSOL the solution of the simulation will be compared to a flow between two infinity long cylinders. Such a geometry is given in figure 8. The flow in this geometry is driven by a radial current between the cylinders and a vertical magnetic field. In this simulation the flow is laminar \( \text{Re} \ll 1 \).

For a flow in this geometry an analytical solution exists [11]. This solution is given by:

\[
J = \frac{A}{r} \quad (39)
\]

\[
U = \frac{1}{2} A H a^2 r \ln r - \frac{B}{2r} + C r \quad (40)
\]

With the following coefficients:

\[
A = J_0 R_0 \quad (41)
\]

\[
B = J_0 R_0^2 H a^2 R_i \ln R_i + \cdots \quad (42)
\]

\[
C = \frac{J_0 R_0 R_i^2 \ln R_0}{2 R_i^2} - \frac{J_0 R_0^2 H a^2 R_i^2 \ln R_i}{2 (R_i^2 - R_0^2)} \quad (43)
\]

The current is set to: \( J_0 = 1.6 \cdot 10^{-4} \) A. This solution is valid in the limit of a thin gap, thus \( (R_0 - R_i) \ll R_i \). The resulting velocity profile is a slightly skewed parabola.

![Figure 8: Geometry with infinity long cylinders](image)
COMSOL makes simulations in finite dimensions, therefore the boundary conditions are chosen in such a way that they match those of the infinite system. This can be done by choosing the same conditions at the vertical walls as were given in section 3.2.2. Thus no-slip for the flow and a potential difference for Poisson’s equation. The horizontal boundary conditions are slip (stress free) for the flow and insulating for Poisson’s equation.

The dimensions of the simulation are:

Inner radius = 10 cm
Channel width = 15 mm
Channel height = 10 mm
The material properties that are used in the simulations are the properties of galinstan:

\[ \mu = 2.4 \cdot 10^{-3} \text{Pa} \cdot \text{s} \]
\[ \rho = 6.440 \cdot 10^3 \text{ kg/m}^3 \]
\[ \sigma = 3.45 \cdot 10^6 \text{ S/m} \]

These material properties will be used for all benchmarks and simulations.

In figure 9 the results from the analytical solution (black lines) and the simulations (red crosses) are plotted. As can be seen the velocity profiles do not change for both the simulations and the analytical solutions. This is due to vertical symmetry of the system that does not allow a side layer, therefore no currents are induced.

For all simulation the difference between the analytical solution and the simulations is in the order of \(1 \cdot 10^{-3}\) compared to a total flow velocity of 1. This can be seen in figure 10.
From the solutions of the analytical solution and the simulation can be concluded that they closely match. This means that numerically framework is complete enough to simulate a liquid metal flow in an axial symmetric system with these boundary conditions. Also it can be concluded that the numerical framework is correctly implemented in COMSOL.

4.2 STRAIGHT SQUARE DUCT FLOW

In this section extra boundary conditions are imposed on the system simulated in the previous section. Both at the bottom and at the top a no-slip condition is imposed on the system. To verify this simulation the solution will be compared to the analytical solution of a flow in an infinitely long straight square duct. This comparison is valid in the limit that the aspect ratio is very small. Therefore a simulation is made of a channel with an inner radius of 100 cm and a channel width and height of 1.5 cm.

![Normalized velocity plots](image)

(a) Hartmann number = 0  
(b) Hartmann number = 1  
(c) Hartmann number = 10  
(d) Hartmann number = 100

Figure 11: Benchmark tests of square duct flow; black line are is the analytic solution whereas the red marks are the simulation.
The geometry of a straight duct is given in figure 4. In chapter 2 the solution for a straight duct flow has been given in equation 16. This solution is valid for a system with insulating boundary conditions.

Because the walls have to be insulating the flow cannot be driven by an applied potential. Therefore the flow is driven by a pressure gradient in the azimuthal direction. This is done by adding a small constant force term in the \( \phi \) direction in equation 36. Thus all boundary conditions of the system are insulating and for the flow no-slip.

In figure 11 the results from the analytical solution (black lines) and the simulations (red crosses) are plotted. In this figure the velocity profiles over a cross section from the inner wall towards the outer wall in the center of the channel have been drawn. As can be seen in figure 12 these profiles match each other within a maximum difference of the order \( 10^{-3} \), compared to a flow velocity of 1. This means that the simulation and the analytical analysis have the same solution.

From the comparison it can be concluded that the simulation completely and correctly calculates the velocities of liquid metals in the
limit of small aspect ratios and laminar flows. This means that the boundary conditions have correctly been implemented.

4.3 ANNULAR DUCT WITH SQUARE CROSS SECTION

In this section the flow in an annular duct with a square cross section with a large aspect ratio will be benchmarked. In this geometry the secondary flows will become important at relatively large dimensionless currents compared to the previous benchmarks.

To validate this flow the results of the simulation will be compared to a simulation made by Vantieghem and Knaepen [19]. They found a numerical solution for liquid metal flows in closed annular channels.

For this benchmark the same simulation is used as in the last section but then with a larger aspect ratio. The channel width is set to 1 meter and the inner radius to 4 meters. The side walls are set to perfect conductors by applying a Dirichlet boundary condition for the potential on either side of the channel. To match the conditions of Vantieghem and Knaepen the magnetic field is set to $1.3 \cdot 10^{-3}$ T and the electric potential to $1.12 \cdot 10^{-7}$ V.

![Graphs](image)

Figure 13: Comparison between the results obtained by Vantieghem and Knaepen and our simulation. The dimensionless parameters are: $Ha = 25$, $Re \approx 100$ and $\zeta = 1/9$ (Ratio of minor radius and major radius). The plotted data are the streamwise velocity profiles along the magnetic field direction. The results are taken at the center of the channel (solid line) and at $1/3$ (circles) and $2/3$ (diamonds) of the channel.
To be able to compare the solutions of the flow the same scaling has to be applied on the velocity fields as Vantieghem and Knaepen have done. The typical velocity is defined as:

\[ U_0 = \frac{V}{B_0 R \log \frac{R+L}{R-L}} \] (44)

with:

- \( R \) = radius at the center of the channel
- \( L \) = half the channel width

In figure 13 the comparison is made. As can be seen in the graphs the flow has the same velocity profile on the cross sections. The reduction of the velocities at different positions in the channel also seems to be the same, as can be seen by the position of the diamonds and circles.

Besides the comparison of the primary flow the secondary flow can also be compared. The secondary flow is defined as:

\[ U_s = \sqrt{u_r^2 + u_z^2} \] (45)

The secondary flow can be scaled, as is done in the paper of Vantieghem and Knaepen, but will not be done here because we are interested in the qualitative behaviour of the secondary flow. In figure 14 the secondary flow profiles are compared. Although it hard to exactly compare the two graphs because they have a slightly different perspective and plotting style. The shape of these figure appears to be largely the same, with three peaks in both figures of the same size. These peaks are formed due to a vortex in the polodial plane. This vortex is thus present in both simulations, and appears to have the same characteristics.

Both simulations show the same behaviour in terms of velocity profiles and secondary flows. This verifies that the simulation produces the correct velocity profiles for both the primary and secondary flow in annular ducts.

From this chapter it can be concluded that the simulations produces correct velocity profiles in axial symmetric geometries. The benchmarked simulations can now be used to determine the scaling behaviour in annular ducts.
Figure 14: Secondary flow comparison between the results obtained by Vantieghem and Knaepen and our simulation. The dimensionless parameters are: $Ha = 25$, $Re \approx 100$ and $\zeta = 1/9$ (Ratio of minor radius and major radius).
MAGNETOHYDRODYNAMIC SCALING
BEHAVIOUR

In the introduction the goal has been stated to develop a simulation to study the scaling behaviour of the velocity flux at surface with the electric and magnetic field in annular channels. To this end a numerical framework has been developed. This framework has been validated, as described in chapter 4. Using this validated simulation the scaling behaviour can be studied.

In chapter 2 three different types of scaling behaviours have been identified. The first area is the linear regime where the scaling of the velocity flux is a linear function of the magnetic field and the applied current. The second regime is the magnetic braking regime where the flow is slowed by the magnetic field. The last regime is the secondary flow regime where the secondary flows induced by the curvature cause a sub-linear scaling of the velocity flux at the surface with the magnetic field and current.

To identify the different areas and to find the values of the dimensionless numbers at which the transition between the areas takes place a simulation has been developed. Using this simulation the flow will be studied. First this is done by studying the behaviour in single annular channels, with vertical magnetic fields. The effects that were observed in the flow will be mentioned and explained. The form in which the results will be presented will also be explained in this section.

After that the complexity of the model is increased by using a magnetic field with radial components. Further complexity is added by adding a fluid layer on top of the channels. The simulations are done in 2D, using an axial symmetric geometry and with the properties of galinstan, see page 26 for the values.
5.1 Individual Annular Channels Channels

5.1.1 Vertical magnetic field

5.1.1.1 Channel with an inner radius of 1 cm

In chapter 2 it has been described that the velocity flux at the surface is expected to scale linearly with the current and the magnetic field, at Hartmann numbers smaller than 1 (see 6, and relatively low flow rates. Besides the change in flow rate also the shape of the flow profile is expected to change. If the Hartmann number is increased the peak velocity is expected to move towards the inner wall. Whereas the peak is expected to move towards the outer wall for higher currents.

Figure 15: Plot of normalized flow profiles for the inner channel at different \(\text{Ha} \) and \(\text{ID} \). The combinations correspond to the extreme values of the Hartmann number and dimensionless current in the simulation and the median value.
To visualise the change in behaviour for different dimensionless currents and Hartmann numbers figure 15 is made from the results of the model. In these figures the normalized velocity profiles of the channel with an inner radius of 1 cm are plotted for both the minimum and maximum current and magnetic field. Also the simulation with the the median values of the simulated magnetic field and potential is plotted. The median value is taken instead of the average because the simulations have a large range with logarithmic equidistant points. These solutions provide a qualitative understanding of the changes a flow gets from changing the magnetic field and potential.

As can be seen in figure 15 the shape of the flow profile changes with changing magnetic field and dimensionless current. Cross sections of the velocity profiles, taken at the free surface ($z = 5 \text{ mm}$), are plotted in figure 16.

In 15d the velocity profile has the peak velocity slightly towards the inner wall of the channel. This same shape can be observed for a slightly larger magnetic field and potential, in 15c.

If either the magnetic field or the potential are increased further the flow profile is deformed. If the magnetic field gets large enough the flow profile starts to shape as a $1/r$-profile. This is seen in 15b,e.

If the potential over the channel is increased to a high value the flow velocity in the channel becomes so large that a circulation in the polodial plane is induced, an effect that is known as the secondary

![Figure 16: Normalized velocity profiles at the top of the flow, for different magnetic fields and electric potentials. The legend corresponds to figure 15. The lines (b) and (e) lie exactly on top of each other](image-url)
flow. The ratio of the secondary flow and the total flow is plotted in figure 17, in this figure the vortex can clearly been seen. This vortex moves momentum from the inner wall towards the outer wall of the channel. The result of this effect on the velocity profile can be seen in 15a. It is on the other hand not observed in 15b. This is because at large Hartmann numbers the magnetic force can suppress the vortex.

The velocity profiles seen in figure 15 give an insight in the dynamics inside the flow, but it does not give information about the scaling behaviour. Therefore a scan is made over the magnetic field and the applied potential for multiple radii. The width of the annular channels is 15 mm. The channel height is 5 mm. Over the channels a potential difference is applied to drive the flow. Therefore the boundary conditions of Poisson’s equation are chosen as described in section 3.2.2 for a cylindrical system. The boundary conditions for the flow are set on the inner, outer and bottom wall to be no-slip. The top wall has a stress free condition.

The flow is simulated for different combinations of the magnetic field and the potential. The range of the simulation is chosen in such a way that all three different scaling regimes can be found. The magnetic field is varied in the range from $1 \times 10^{-4}$ to $1 \times 10^{-1} \text{T}$. The values are logarithmically equidistantly spaced in 31 steps. The potential is varied between a value of $1 \times 10^{-12}$ to $2.5 \times 10^{-8} \text{VT}$ for the product of the potential and magnetic field. The potential has 45 logarithmically equidistant points. The velocity flux at the surface of a simulation with this scan for the a channel with an inner radius of 1 cm is plotted in figure 18.

From figure 18 it is hard to see transition between the areas of different scaling. A difference in scaling should become apparent in the

![Image](image_url)
Figure 18: Scatter plot of the dimensionless velocity flux through an annular channel with inner radius of 1 cm for various Hartmann numbers and dimensionless currents.

graphs as a change in gradient. To better see this transition figure 19 is made. In this plot the dimensionless velocity flux at the surface rate divided by the product of the dimensionless current and the Hartmann number is plotted versus the dimensionless current and the Hartmann number.

A constant colour in figure 19 means linear scaling behaviour. A gradient in the colours means different scaling, in this particular case

Figure 19: Velocity flux at the surface divided by the product of the Hartmann number and the dimensionless current for the inner channel. The cross sections (dotted lines) are plotted in figure 20. The solid black is the transition between the areas of different scaling.
sub-linear scaling. From this figure two cross sections are made. In
the first cross section a scan is made over the magnetic field with a
constant current, in figure 19 the gray and white dotted line corre-
sponds to this cross section. The plot of this cross section can be seen
in figure 20a. A second sweep is made over the dimensionless current
with a constant magnetic field, that is the vertical black and white dot-
ted line. This is plotted in figure 20b. The same cross sections are also
made in figure 18 and plotted as images 20c and d.

From the plots in figure 20 the scaling of the velocity flux can be
obtained. First the scaling of the magnetic field can be seen, graph
20a. A flat profile in this graph means linear scaling. As can be seen
starting from a Hartmann number of about 1 the function starts to
decrease, and thus scales not linear anymore. In the plot 20b for the
velocity flux versus the current similar behaviour is observed. Start-
ing from a dimensionless current of about 0.01 the velocity flux starts
to decrease, and the scaling is not linear anymore. The non-linear scal-
ing can also be observed in the plots of the velocity flux 20c and d,
in these plots the gradients decrease when non-linear scaling is ob-
served in plots 20a and b.
A condition for the linearity of the scaling has been made to compare simulations with each other. The velocity flux divided by the product of the current and magnetic field does not scale linearly anymore when it is smaller than a certain threshold value. The algorithm analysing the simulations uses the logarithm of the velocity flux, the threshold value is set to 96% of the maximum value of the logarithm of the velocity flux at the surface, thus mathematically:

\[
\frac{\log(v)}{\max(\log(v))} \begin{cases} 
> 0.96, \text{ linear} \\
< 0.96, \text{ non-linear}
\end{cases}
\] (46)

Or in the non logarithmic case:

\[
\frac{v}{\max(v)} \begin{cases} 
> 0.912, \text{ linear} \\
< 0.912, \text{ non-linear}
\end{cases}
\] (47)

This value was chosen because from this point it was obvious that the scaling was not linear anymore. The output data has a fairly poor resolution, therefore a first order interpolation is used to determine the threshold values more exactly. This can be done since the flux is always monotonically decreasing in the transition area with respect to increasing Hartmann number or dimensionless current.

The result for this threshold is plotted in figure 19 as a black line. This line is the boundary between linear and non-linear scaling. The area to the left and below the line is the area that scales linearly. To the right of this curve the magnetic force slows the fluid down. Non-linear scaling above the line is due to the secondary flow induced by the curvature of the channel.

The scaling of the function can be obtained from the rate of change in figure 18. To determine the scaling the following function is assumed between to successive points in the plot:

\[ y = b \cdot x^a \] (48)

In this case \( y \) is the dimensionless velocity flux, \( b \) is a scaling constant, \( x \) is either the Hartmann number or the dimensionless current, and finally \( a \) is the power to which the function scales. From this function it is possible to determine the power \( a \). This is done using the following calculation: First the ratio between two points is taken:

\[ \frac{y_2}{y_1} = \frac{b \cdot x_2^a}{b \cdot x_1^a} \] (49)

And from this the power \( a \) can easily be obtained:

\[ a = \frac{\ln \left( \frac{y_2}{y_1} \right)}{\ln \left( \frac{x_2}{x_1} \right)} \] (50)
This function is used to find the power $a$ with respect to both the Hartmann number or the dimensionless current. If $a$ is 1 it means that the function scales linearly. If the function has a power smaller than 1 it means that the slope is less than linear. A value below 0 means that the function starts to decrease.

In figure 21 the power $a$ is plotted for both for scaling with the Hartmann number and the dimensionless current. The following scaling behaviour can be observed from figure 21:
• Linear scaling for small Hartmann number and dimensionless current
• Sub-linear scaling for large dimensionless currents and small Hartmann numbers
• Sub-linear scaling for large Hartmann numbers

This scaling is in agreement with what was expected in chapter 2.

5.1.1.2 Channel with larger inner radii

For a channel with an inner radius of 1 cm, the scaling behaviour has now been analysed. To see what the effect is of the curvature of a channel, simulations have been made with different radii. To be able to properly compare the boundaries of mains with different scaling a factor is added to the dimensionless current, the aspect ratio:

\[ I_{D, aspect} = I_{\mu_0} \sqrt{\frac{\mu}{\mu_0}} \cdot \frac{a}{R} \] (51)

This extra factor compensates for the difference in the area that is driven by the electromagnetic forces, and the driven area is proportional to the radius of the channel.

![Graph showing boundary lines for linear scaling for channels with larger inner radii](image)

Figure 22: Comparison of the boundary lines for linear scaling for channels without atop layer with a vertical magnetic field.
In figure 22 the boundary between linear and non-linear scaling is plotted for an inner channel radius of 1, 10 and 100 cm. The scaling has for all channels a very similar shape. For the magnetic scaling behaviour a vertical boundary is found, at nearly the same place, around $H_a = 2$. The boundary for non-linear scaling due to the curvature is found on lines of constant $J \times B$. The lines differ for different channel diameters. For larger diameters of the channels the linear scaling is observed for higher dimensionless currents. This is also what was expected, as the circulation is induced by the curvature of the channel. This effect happens more easily in channels with smaller diameters, reducing the velocity. This results in non-linear scaling at lower currents.

5.1.2  Realistic magnetic field

In the previous section a method has been introduced to characterize the areas of different scaling in individual annular channels with a vertical magnetic field. In this section the effects of a realistic magnetic field are studied. To do this the magnetic fields module is added to the simulation to facilitate the calculation of a more realistic magnetic field.

The realistic magnetic field consists of a vertical magnetic field with radial components. Such a magnetic field can, in an experiment, be created using bar magnets in a grid layout as in figure 23. Such a geometry allows to have a magnetic field which is highly symmetric in axial direction. The magnetic field is simulated in COMSOL using the geometry of figure 23.

Figure 23: Top view of the grid that is simulated of a grid for bar magnets and the position of the channels
The boundary conditions for the simulated geometry are given in section 3.2.2 for cylindrical coordinates. The magnets are placed at a center radius of 27.5, 42.5, 67.5 and 92.5 mm. In order to create the same field strength as in the experiment the magnets have been given a remanent flux density of $1.73 \hat{e}_z \, \text{T}$. A plot of the magnetic field strength in the middle channel is plotted in figure 27.

To change the magnetic field the position of the magnets is changed from 400 mm to 150 mm measured from the bottom of the channel. This corresponds (for the middle channel) to a change in Hartmann number of 0.097 to 2.44. The potential over the channels is changed in order to create a change in the dimensionless current from $5.6 \cdot 10^{-5}$ to $3.7 \cdot 10^{-2}$. For three different inner radii the channels are simulated. These radii are the same as in the experimental set-up (10, 35 and 60 mm). In this range the transition between the regimes of different scaling can be observed.

The computation of the flow in a realistic magnetic field is computationally quite a bit more challenging. Therefore the resolution in the Hartmann number and the dimensionless current is lower than in the calculation only using a vertical field. The position of the magnets is increased linearly in steps of 25 mm. The potential is increased in 11 logarithmically equidistant steps.
Figure 25: Plot of normalized flow profiles vs magnetic field and potential for the inner channel with a realistic magnetic field.

Figure 26: Comparison of the contour lines for linear scaling for a simulation with a realistic field and a vertical field. Dashed lines: vertical field; Solid lines: realistic field. The diamonds give the positions of the velocity profiles in figure 25.
To compare the change of the velocity flux at the surface for the realistic field with the vertical field two plots are made. The first one is the plot of the velocity profiles. In figure 25 the change of behaviour inside the flow can be compared. In figure 26 the boundary of linear scaling are plotted for the solutions of the both simulations. The channels for both the realistic and the vertical field have the same radii as in the experiment.

In figure 25 the normalized flow profile can be seen. The same change in behaviour can be observed as in figure 15, but is in this case less pronounced. This is because the plots are made in a smaller range for the Hartmann number and dimensionless current. The positions of the velocity profiles plotted in figure 25 are plotted as diamonds in figure 26.

In figure 26 the boundaries for linear scaling have been plotted. It can be observed that in the simulations with a more realistic magnetic field the non-linear scaling starts at lower Hartmann numbers and dimensionless currents. This can be contributed to the inhomogeneity of the field within the channel for realistic fields. The magnetic field is the largest at the bottom of the channel and decreases towards the top. This decreases the maximum velocity gradient possible at the bottom of the channel. Since the gradient at the bottom determines the velocity flux at the surface, a inhomogeneity at the magnetic field will cause a lower velocity flux.
In figure 27 a plot is made of the relation between the position of the magnets and the magnetic field, and its minimum and maximum value within the middle channel. The maximum difference is 6% in the channels. The difference in the magnetic field is the largest for high Hartmann numbers, therefore the velocity flux at the surface will relatively be smaller for large Hartmann numbers. This causes for non-linear scaling at lower Hartmann numbers than with a vertical field, and the boundary for non-linear scaling will thus move to the left.

The curves also move downwards and are also closer together than for the vertical field, and the order of the curves is swapped in vertical direction. This swapping in vertical direction could be an effect of the low resolution of the simulation and limited range. These lines can cross each other for lower Hartmann number and higher dimensionless currents. Therefore no conclusion can be made for the exact order of the lines in the range for relatively high dimensionless currents.

The boundary line for linear scaling moves downward because either the interaction between the secondary flow and the inhomogeneous magnetic field is stronger or the the inhomogeneous magnetic field induces stronger secondary flows. The exact reason for this could not be determined.

The conclusion that can be drawn from the simulation of the flow in a single annular channel with a magnetic field with vertical and radial components is that the regime for linear scaling is smaller. At lower Hartmann numbers the scaling velocity flux at the surface becomes sub-linear because of the inhomogeneity of the magnetic field. The flow also starts to show sub-linear scaling for lower dimensionless currents, this is caused by an increased secondary flow.
5.2 GEOMETRY WITH A TOP LAYER

5.2.1 vertical magnetic field

In this section the same simulation is used as in section 5.1.1.1, only the geometry of the channels has been changed to the geometry of the experiment by adding a top layer.

As in the previous sections two plots are made, the first one gives the velocity profiles. In this figure the change in behaviour of the flow can be seen. In the second figure 29 the boundaries of linear scaling have been plotted.

In figure 28 the velocity profiles are plotted in the three identified domains. In plot a the deformation of the velocity profile can be seen due to the high velocity. It can be seen that the maximum velocity is moved towards the outer wall and that the secondary flow moves momentum inwards. This deformation is caused by a combination of the curvature of the channel and the interaction between the secondary flow and the geometry. These two effects cause sub-linear scaling for

![Figure 28](image1.png)

(a) $H_a = 2.84 \cdot 10^{-2}$, $I_D = 1.22$

![Figure 28](image2.png)

(b) $H_a = 2.84 \cdot 10^{-2}$, $I_D = 9.96 \cdot 10^{-5}$

![Figure 28](image3.png)

(c) $H_a = 2.84$, $I_D = 6.49 \cdot 10^{-6}$

Figure 28: Plot of normalized flow profiles vs magnetic field and potential for the experimental setup with the inner channel driven by an electric potential with a vertical magnetic field.
a lower product of the Hartmann number and the dimensionless currents. In figure b were neither magnetic braking or curvature effects. In this regime the flow scales linearly. In figure c magnetic braking dominates the flow, in the bulk layer on top the magnetic field also slows down the flow. This results in a small change in the velocity profile, the profile is slightly more leveled in plot c than in b. This causes non-linear scaling for the velocity flux at the surface, an effect that can also be observed in figure 29. In this figure the boundary for linear scaling moves to the left, i.e. non-linear scaling happens already for smaller Hartmann numbers.

From these results it can be concluded that the geometry reduces the regime of linear scaling both from the magnetic braking as the secondary flow side. The shape of this area stays the same.

The method used to simulate these flows does not yield errors in the simulated domain. However when the current is increased problems can occur. This problem arises because it is difficult for COMSOL to compute accurate boundary fluxes in diffusion dominated problems. The increase the domain which simulation can be made the corners have been rounded with a radius of 0.5 mm in the simulation. This allowed for consistent calculations, but still in a relativity
small range. The problem can be fully be solved by using the so called weak constraints in COMSOL, for more details on this problems see the COMSOL knowledge base [1].

5.2.2 realistic magnetic field

In this section the geometry of the previous section will be implemented in the simulation of section 5.1.2. In figure 30 the normalized flow profiles for the different regimes have been plotted. For computational and stability reasons the simulated domain is smaller than in the previous simulations. The simulation for the channel with an inner radius of 35 mm could not be calculated at Hartmann numbers in which the transition for magnetic braking was visible, because the simulation did not converge in that regime. However using the other two radii all the three regimes could be identified.

The flow profiles in figure 30 do not change as much as in figure 28, this is because the simulated domain is much smaller. This can be seen by comparing the positions of the diamonds in figure 31 and 29. But still the same qualitative changes in the flow can be observed by

![Flow profiles](image)

(a) \( H_a = 9.21 \cdot 10^{-2}, \quad I_D = 6.21 \cdot 10^{-2} \)

(b) \( H_a = 9.21 \cdot 10^{-2}, \quad I_D = 1.10 \cdot 10^{-4} \)

(c) \( H_a = 1.14, \quad I_D = 9.49 \cdot 10^{-5} \)

Figure 30: Plot of normalized flow profiles vs magnetic field and potential for the experimental setup with the inner channel driven by an electric potential with a vertical magnetic field.
looking carefully at the results from the simulation. In figure (a) it is barely visible but the flow moves outwards due to the curvature. In figure (b) the linear solution can be seen. And, finally in figure (c) the profile flattens due to the magnetic forces.

The resulting plot for the boundary of the linear scaling is plotted in figure 3.1. Due to the inhomogeneity of the magnetic field, the regime of linear scaling is smaller both from the side of magnetic braking as from the side of the secondary flows.

The transition between linear regime and the magnetic braking regime takes place at different Hartmann number. This difference can be contributed to the way the Hartmann number is obtained. The magnetic field that is used to calculate the Hartmann number is obtained by taking the average over the entire container. In the container the difference in magnetic field can be up to 10%. The boundary for non-linear scaling is therefore moved to the right for the inner channel, and to the left for the outer channel.

From this section it can be concluded that for the experimental set-up non-linear scaling will start at smaller Hartmann numbers and di-
mensionless currents, than what would be expected from strait ducts or closed annular channels.
6.1 Conclusion

In this thesis the influence of a top layer on scaling behaviour of the velocity flux at the surface of a liquid metal flow with the Hartmann number and dimensionless current was studied in annular channels. The following conclusion can be drawn from the work:

A simulation, using the magnetohydrodynamic theory, has been developed that can simulate a flow of a liquid metal in an arbitrary geometry. This model has been validated in axial symmetric channels using benchmarks from literature. With this simulation it is possible to study the influence of top layer on the scaling behaviour of the velocity flux at the surface with the magnetic field and the electric current.

This simulation, in combination with a theoretical analysis, has first been used to identify three areas of different scaling in a single annular channel. It was found that for small Hartmann numbers and dimensionless currents the velocity flux at the surface scales linearly with the electric current and magnetic field. If either the Hartmann number or the dimensionless current is increased it was found that the flow velocity flux at the surface does not scale linearly anymore.

The velocity flux at the surface does not scale linearly with increasing Hartmann number because at large Hartmann numbers. This because currents that are induced by the flow, of a metal, through a magnetic field cause a braking force in the flow. This effect starts in individual channels without a top layer to dominate the flow for a Hartmann number of about two. The start of this effect is independent of the inner radius of a channel.

If the product of the magnetic field and the current becomes too large at small Hartmann numbers secondary flows begin to influence the velocity flux at the surface. These secondary flows are induced by the curvature of the channels. The scaling of the velocity flux is reduced to sub-linear above certain values of the product of the Hartmann number and the dimensionless currents. This value depends on the radius of the channel: From a value of $5.5 \cdot 10^{-3}$ AT for an inner radius of 1 cm to $3.2 \cdot 10^{-2}$ AT for a channel with a inner radius of 100 cm, whereby the dimensionless current is adjusted for the differ-
ence in aspect ratio.

If a top layer is added to the channels, both the current and the flow are changed. However in this geometry the same three areas are observed. The place where the transition between these areas takes place is changed. Magnetic braking becomes dominant for lower Hartmann numbers, it happens for about half the Hartmann number compared to an individual channel. The non-linear scaling of the velocity flux at the surface at lower Hartmann number occurs because also in the top layer currents are induced, and just as in the channels they cause braking of the flow.

The influence of the secondary flows becomes also more dominant at lower values of the product of the Hartmann number and dimensionless current. The change is larger for the outer channel than for the inner channel. The product of the dimensionless current and the Hartmann number for which the change takes place is about three times lower for the outer channel and two times lower for the inner channel, again adjusted for the difference in aspect ratio.

Besides the main question the additional question was: How does the scaling of the velocity flux at the surface change if the magnetic field is not perfectly parallel and perpendicular to the surface. To answer this a simulation was made in which the magnetic field was made using bar magnets. From the results of the simulation it was found that a non-homogeneous magnetic field does significantly influence the scaling with the Hartmann number and dimensionless current, both in individual channels and in the system with the top layer.

For both the individual channel and the system with the top layer the same changes to the scaling were found. Magnetic braking occurred for much lower Hartmann numbers, the change was in individual channels from 2 to around 0.5. The start of the transition between the linear scaling and the magnetic braking does now depend on the radius. This is because of the difference in homogeneity of the magnetic field in the channels.

For the system with the experimental set-up a very similar change is seen from about 1 to 0.4. The reason that the magnetic braking happens at a lower Hartmann number is due to the fact that the magnetic field is not homogeneous anymore, it is strongest at the bottom of the channel. Therefore the maximum gradient of the velocity profile is limited more in the bottom of the channel than it would be with a parallel magnetic field, and thus the resulting velocity is lower.
The secondary flow also becomes more important at lower value of the product of the magnetic field and the dimensionless current. Therefore there appears to be a coupling between the non-homogeneous magnetic field and the secondary flow.

6.2 DISCUSSION

The simulations were made in the idealized case, in reality other effects come into play, due slightly different geometry and boundary conditions, also the range can be extended to fully incorporate the potential experimental regime.

The simulated geometry of the experimental set-up is slightly changed from the real set-up. The corners were rounded to ensure a correct flux calculation of the potential. This could have an influence on the calculated flow as the secondary flow has to follow a surface without sharp corners in the simulation opposed to the experiment. Another solution to this problem to used the so-called weak constraints in COMSOL for the calculation of the boundary fluxes. The weak constraints would also allow to extend the range in which simulations can be made to higher Hartmann numbers and dimensionless currents.

The boundary conditions that are used to simulate the current restricted the simulation to perfectly conducting boundary conditions for the walls where the potential is applied and insulating for the other walls. These conditions are not necessarily met in experimental set-ups, especially not the perfectly conducting walls. In a channel with walls with a finite conductivity the volumetric rate is higher than through a channel with a perfectly conductive wall.

Due to the choice of the modules in COMSOL the simulations had a limited regime in which they could be preformed. The model is limited in the range of the driving terms (current and magnetic field), if they become too large the flow can become unstable this can either be due to the development of turbulence or due to numerical instabilities. The simulation has been developed to cope with turbulence. To extend the simulation to the flow in this regime the turbulent flow module can be used in COMSOL.

It was found from the simulations that there was a coupling between the secondary flow and the inhomogeneous magnetic field. But no reason has been found to what caused this. A possible explanation could be that the gradient of the magnetic field increase the pressure gradient that is needed to keep the flow following the curved channel. This pressure gradient causes the secondary flow, a coupling between
the two gradients could act as an initiator of a stronger secondary flow.

6.2.1 Comparison with other MHD simulation codes

The simulation described in this report has been developed to simulated liquid metal flows in COMSOL. The model is validated by a comparison with two analytic solutions and a model published by Vantieghem and Knaepen. These benchmarks showed that the simulation is capable of correctly calculating velocity fields of laminar liquid metal flows. The simulation can do this in an arbitrary geometries (as long as enough computer power is available). In the simulation extra physics can easily be implemented by adding other modules in COMSOL.

Besides the simulation that we developed other codes have been developed to simulated MHD-flows. The two most well known codes are: HIMAG and YALES2.

HIMAG is a code developed at UCLA, and it is currently being distributed by a company called Hypercomp [9]. UCLA’s goal was to design a liquid lithium wall for a fusion reactor. HIMAG is used to simulate this liquid wall. The code is designed to simulate complex free surface liquid metal MHD flows. The algorithm allows: any geometry; high Hartmann numbers; thin Hartmann layers and parallel solving. Their work differed from the simulation presented in this report, as they focussed on higher Hartmann numbers and deformable free surfaces (e.g. waves). This allows for simulations that are better suited for calculations in a tokamak. The main reason for this is that the magnetic fields are strong in tokamaks and therefore the Hartmann numbers are typically high. However no calculations have been published using HIMAG in a comparable geometry to ours and with similar Hartmann numbers.

YALES2 is another code that is available and capable of solving the MHD-equations [19]. YALES2 has originally been developed by Coria to simulated two phase combustion (e.g. for simulations of engines). YALES2 also has a library that is capable of solving the MHD-equation. This has been used by Vantieghem and Knaepen to calculated a liquid metal flow in various geometries. One of these geometries was a closed annular duct with square cross section. In chapter 4 these results were used to validated our simulation. Their work mainly focused on flows with high Hartmann numbers and Reynolds numbers, to study magnetic suppression of turbulence.
Besides these stand alone MHD-codes there have also been a number of attempts to simulated a liquid metal flow in COMSOL. This has been done in the group of Gutierrez-Miravete at department of Engineering and Science at the Renseale ar polytechnic institute in Hartford. The goal of these simulations was to simulated heat transfer in liquid metal flow for industrial applications in COMSOL. To do this both simulations of a flow between parallel plates [4] and over a backward step [20] were made in COMSOL. They showed that COMSOL simulations can accurately reproduce theoretical expectations of liquid metal flows, but are less advanced than the simulations explained in this report. Their simulations were only used to simulate liquid metal flows in simple geometries with vertical magnetic fields. Whereas our simulations can use much more complex geometries and boundary conditions.

The main difference between HIMAG, YALES2 and our simulation is that COMSOL allows to add other physics. With this flexibility simulations of combined physical phenomena can be made. This allows for the use of our code in simulations that are relevant for liquid metal flows in nuclear fusion applications. Examples of physics that can be added is the influence of the plasma on the liquid metal or the response of a liquid metal to transient heat loads of ELMs.

6.3 FUTURE DIRECTIONS

6.3.1 Simulations

The simulations have been used to find the boundary lines for the different scaling regimes. But the simulations are able to do much more than that. As a first step the simulations can be used to predict the results of the experimental set-up. Making a simulation with the same boundary conditions as the experiment can be used to validate the simulation in a range in which it has not yet been validated. To fully eclipse the experimental range the simulation has to be expanded with the incorporation of the weak constraints to enforce a correct calculation of the boundary fluxes at large Hartmann numbers. If this does not expand the operational domain of the simulation enough, because the flow is turbulent, the turbulent flow module of COMSOL might need to be used.

Also simulations can be made in different geometries. A possible good starting point would be to simulate geometries in which already relevant experiments and simulations have been done. An example of this would be the MTOR experiment of Princeton [10], this experiment has also been simulated using HIMAG. In this experiment a flow of galinstan on an inclined plane in strong magnetic fields
has been studied. This experiment has been done to study the behaviour of liquid metal flow for divertor purposes. A positive match between this experiment and the simulations would extend the domain in which the simulation is benchmarked into the domain where it is relevant for fusion applications.

6.3.2 Liquid metal divertor

Two possible mechanisms can be used to transport the heat out of a liquid metal divertor: conduction and convection. The liquid metal layer acts, in the case of conduction, as a protection layer for the divertor. The heat is transported through the liquid metal layer and the heat exchanger below it by means of conduction. The amount of heat that is conducted through such a layer can be calculated using Fourier’s law \( q_x = \lambda \frac{\partial T}{\partial x} \). From this relation, it can be seen that the amount of heat conduction is proportional to the temperature difference. The temperature at the surface is in this case the limiting factor. This is because liquid metals have a maximum temperature before they start to evaporate too fast. The flow of a liquid metal is only needed to refresh the material. The material must be refreshed because the material slowly evaporates. The flow velocities are smaller than for which our simulation is designed.

The second mechanism to transport heat is through convection. In this case the flow transports the heat out of the divertor. In this case the heat load is still limited by the maximum temperature of the liquid metal, but not solely through Fourier’s law. The temperature increase is also determined by the heat capacity of the metal and the time the metal is exposed to the heat flux. This time can be limited by a fast flow. In this type of liquid metal divertor it could be useful to have an active control.

As mentioned in the introduction local control of the flow in the divertor could be used to increase the self-healing capability of the divertor, but that is nothing new. What our results show is that it is now known when the liquid metal in the divertor is in certain scaling regimes. To be able to change between the linear scaling regime and the other regimes, the Hartmann number should be below one. Typical values in a tokamak are a magnetic field of about 5 T, in the calculation of the Hartmann number the perpendicular component to the flow is important. Since the surface of the divertor typically makes a very small angle with the magnetic field lines, this field is easily reduced by a factor of 10, leaving an effective field left of 0.5 T. To get a Hartmann number of one the channel width for galinstan is 50 \( \mu \text{m} \). This is too small to be useful, thus therefore the flow is always in the regime of magnetic braking. The channel width for lithium would
even be lower as is a better conductor and less viscous than galinstan.

It could be beneficial to have a secondary flow in the divertor, e.g. it could for example lower the recycling. The recycling is improved because the material will saturate slower. To enforce a secondary flow at large Hartmann numbers a larger current is needed than to drive the flow without a secondary flow. In the simulated regime it was observed that the magnetic field suppressed the secondary flow. Therefore no criteria has been found for the transition between the magnetic braking area and the secondary flow area. But it can be concluded from the simulations that the boundary is for larger products of the dimensionless currents and Hartmann numbers than used in the simulation.

Realistically the liquid metal divertor will be in the regime of magnetic braking, a secondary flow might be induced by using large currents.

Our work has also generated some generic results which might have applications beyond the scope of nuclear fusion. An example is the use of the stress free surface of the liquid metal for mirror applications. These mirrors made with liquid metals allow for very flat and reflective mirrors. A example of such a mirror is a liquid metal telescope, a round bath is rotated to create a parabolic mirror. The simulations can be used to calculate if it is possible to make a parabolic mirror with a set-up like the one that was used in the experiment. Benefits of such a system would be that there are no moving parts resulting in less vibrations.
The Buckingham’s \( \Pi \) theorem is a procedure to create a complete set of independent dimensionless variables of a problem. This set can be used to see how a problem scales. In this sense it is used for the flow in annular channels. In the simulations the scaling behaviour of the velocity flux will be studied. The dimensionless parameters of this problem will be derived in this chapter.

The velocity flux is a function of:

\[
Q_s = f( v, \rho, \mu, a, I, B, \sigma, \mu_m )
\]  

These variables have the following dimensions:

\[
\begin{align*}
[Q_s] &= L^3 T^{-1}; [v] = LT^{-1}; [\rho] = ML^{-3}; \\
[\mu] &= ML^{-1} T^{-1}; [a] = L; [I] = I; \\
[B] &= I^{-1} MT^{-2}; [\sigma] = M^{-1} L^{-3} T^3 I^2; \\
[\mu_m] &= MLT^{-2} I^{-2}
\end{align*}
\]

There are nine variables and four physical quantities (length, time, mass and current). According to Buckingham’s \( \Pi \) theorem there are \((9 - 4) = 5\) independent dimensionless numbers. These numbers can be obtained from this function:

\[
\Pi = Q_s^{a_1} v^{a_2} \rho^{a_3} \mu^{a_4} a^{a_5} I^{a_6} B^{a_7} \sigma^{a_8} \mu_m^{a_9}
\]  

To make \( \Pi \) dimensionless the parameters \( a \) have to be chosen correctly. This restriction leads to the following equation for the base dimensions:

\[
\begin{align*}
M: & \quad a_3 + a_4 + a_7 - a_8 + a_9 = 0 \quad (55) \\
L: & \quad 3a_1 + a_2 - 3a_3 - a_4 + a_5 - 3a_8 + a_9 = 0 \quad (56) \\
T: & \quad -a_1 - a_2 - a_4 - 2a_7 + 3a_8 - 2a_9 = 0 \quad (57) \\
I: & \quad a_6 - a_7 + 2a_8 - 2a_9 = 0 \quad (58)
\end{align*}
\]

As repeating variable \(( \mu, a, \sigma, \mu_m )\) are chosen. Since that of the velocity flux the relations want to be know, \( a_1 \) is chosen to be \( 1 \). Therefore \( a_2, a_3, a_6 \) and \( a_7 \) are chosen to be \( 0 \). Thus the previous relations simplify to:
\[ \Pi_1 = \mu \alpha^4 \alpha^5 \beta \alpha^6 \sigma \alpha^8 \mu \alpha^9 \] (59)

M: \( a4 - a8 + a9 = 0 \) (60)
L: \( 3 - a4 + a5 - 3a8 + a9 = 0 \) (61)
T: \( -1 - a4 + 3a8 - 2a9 = 0 \) (62)
I: \( 2a8 - 2a9 = 0 \) (63)

This system can be solved to: \( a1 = 1, a4 = 0, a5 = -1, a8 = 1 \) and \( a9 = 1 \). Resulting a dimensionless number \( \Pi_1 \):

\[ \Pi_1 = \frac{Q_s \sigma \mu m}{\alpha} \] (64)

For the dimensionless velocity \( a2 \) is chosen to be 1. Therefore \( a1, a3, a6 \) and \( a7 \) are chosen to be 0. Thus the previous relations simplify to:

\[ \Pi_2 = v \alpha^2 \mu \alpha^4 \alpha^5 \sigma \alpha^8 \mu \alpha^9 \] (65)

M: \( a4 - a8 + a9 = 0 \) (66)
L: \( a2 - a4 + a5 - 3a8 + a9 = 0 \) (67)
T: \( -1 - a4 + 3a8 - 2a9 = 0 \) (68)
I: \( 2a8 - 2a9 = 0 \) (69)

This system can be solved to: \( a2 = 1, a4 = 0, a5 = 1, a8 = 1 \) and \( a9 = 1 \). Resulting a dimensionless number \( \Pi_2 \):

\[ \Pi_2 = v \alpha \sigma \mu m \] (70)

For the dimensionless density \( a3 \) is chosen to be 1. Therefore \( a1, a2, a6 \) and \( a7 \) are chosen to be 0. Thus the previous relations simplify to:

\[ \Pi_3 = \rho \alpha^3 \beta \alpha^4 \alpha^5 \sigma \alpha^8 \mu \alpha^9 \] (71)

M: \( 1 + a4 - a8 + a9 = 0 \) (72)
L: \( -3 - a4 + a5 - 3a8 + a9 = 0 \) (73)
T: \( -a4 + 3a8 - 2a9 = 0 \) (74)
I: \( 2a8 - 2a9 = 0 \) (75)

This system can be solved to: \( a3 = 1, a4 = -1, a5 = 0, a8 = -1 \) and \( a9 = -1 \). Resulting a dimensionless number \( \Pi_3 \):

\[ \Pi_3 = \frac{\rho}{\mu \sigma m} \] (76)

For the dimensionless current \( a6 \) is chosen to be 1. Therefore \( a1, a2, a3 \) and \( a7 \) are chosen to be 0. Thus the previous relations simplify to:
Here is the text in plain text format:

\[ \Pi_4 = \mu a^4 a^5 I a^6 o a^8 \mu_m a^9 \]  \hspace{1cm} (77)

M: \[ a^4 - a^8 + a^9 = 0 \]  \hspace{1cm} (78)

L: \[ - a^4 + a^5 - 3a^8 + a^9 = 0 \]  \hspace{1cm} (79)

T: \[ - a^4 + 3a^8 - 2a^9 = 0 \]  \hspace{1cm} (80)

I: \[ 1 + 2a^8 - 2a^9 = 0 \]  \hspace{1cm} (81)

This system can be solved to: \( a_4 = -\frac{1}{2}, a_5 = 0, a_6 = 1, a_8 = \frac{1}{2} \) and \( a_9 = 1 \). Resulting a dimensionless number \( \Pi_5 \):

\[ \Pi_4 = I \mu_m \sqrt{\frac{\sigma}{\mu}} \]  \hspace{1cm} (82)

For the dimensionless magnetic field \( a_7 \) is chosen to be 1. Therefore \( a_1, a_2, a_3 \) and \( a_6 \) are chosen to be 0. Thus the previous relations simplify to:

\[ \Pi_5 = \mu a^4 a^5 B a^7 o a^8 \mu_m a^9 \]  \hspace{1cm} (83)

M: \[ a^4 + 1 - a^8 + a^9 = 0 \]  \hspace{1cm} (84)

L: \[ - a^4 + a^5 - 3a^8 + a^9 = 0 \]  \hspace{1cm} (85)

T: \[ - a^4 - 2 + 3a^8 - 2a^9 = 0 \]  \hspace{1cm} (86)

I: \[ - 1 + 2a^8 - 2a^9 = 0 \]  \hspace{1cm} (87)

This system can be solved to: \( a_4 = -\frac{1}{2}, a_5 = 1, a_7 = 1, a_8 = \frac{1}{2} \) and \( a_9 = 0 \). Resulting a dimensionless number \( \Pi_5 \):

\[ \Pi_5 = B a \sqrt{\frac{\sigma}{\mu}} \]  \hspace{1cm} (88)

This number is known in the literature as the Hartmann number (\( H_a \)).


