Electromagnetic effects in cylindrical pipe flow
Electromagnetic effects in cylindrical pipe flow

PROEFSCHRIFT

ter verkrijging van de graad van doctor
aan de Technische Universiteit Delft
op gezag van de Rector Magnificus, prof.dr.ir. J. Blaauwendraad,
in het openbaar te verdedigen ten overstaan van een commissie,
door het College van Dekanen aangewezen,
op vrijdag 19 december 1997 te 10.30 uur

door

Bendiks Jan BOERSMA

Werktuigkundig ingenieur

geboren te IJlst
Dit proefschrift is goedgekeurd door de promotoren:

prof.dr.ir. F.T.M. Nieuwstadt
prof.dr. F.C. Schüller

Samenstelling promotiecommissie:

Rector Magnificus, voorzitter
Prof.dr.ir. F.T.M. Nieuwstadt, Technische Universiteit Delft, promotor
Prof.dr. F.C. Schüller, Universiteit Utrecht, promotor
Prof.dr.ir. J.J.H. Brouwers, Universiteit Twente
Prof.dr.dipl-ing. K. Hanjalic, Technische Universiteit Delft
Prof.dr.ir. A.A. van Steenhoven, Technische Universiteit Eindhoven
Prof.dr.ir. P. Wesseling, Technische Universiteit Delft
Dr.ir. A.J. van Bekkum, Krohne-Altometer, Sliedrecht

Published and distributed by:

Delft University Press
Mekelweg 4
2628 CD Delft
The Netherlands
Telephone: +31 15 2783254
Fax: +31 15 2781661
E-mail: DUP@DUP.TUDelft.NL

ISBN 90-407-1530-0/CIP

Copyright ©1997 by B.J.Boersma

All rights reserved. No part of the material protected by this copyright notice may be reproduced or utilized in any form or by any means, electronic or mechanical, including photocopying, recording or by any storage and retrieval system, without permission from the publisher: Delft University Press.

Printed in the Netherlands
Voorwoord

Voor u ligt het resultaat van vier jaar onderzoek, uitgevoerd aan het Laboratorium voor Aëro en Hydrodynamica van de TU-Delft. Dit onderzoek was niet mogelijk geweest zonder de hulp van vele collega's op het lab. Enkele van hen wil ik hier graag met name noemen. Natuurlijk mijn eerste promotor Frans Nieuwstadt die me de vrijheid heeft geboden om allerlei zijwegen in te slaan en het promotie onderzoek als een kapstok te gebruiken. Jack Eggels en Mathieu Pourquié zonder wiens voorafgaande (pro)meerwerk het niet mogelijk was geweest dit proefschrift te schrijven. Graag wil ik ook graag “mijn” studenten, Bas van Haarlem, Martijn Nass en Edwin de Korte bedanken voor de prettige samenwerking.

Mountain View, CA, oktober '97

Dit onderzoek werd gefinancierd door de Stichting Technische Wetenschappen (STW), onder project nr. D'TN22.2646. De berekeningen waarvan de resultaten in dit proefschrift zijn gepresenteerd zijn hoofdzakelijk uitgevoerd op de CRAY-C90 van de Stichting Academisch Rekencentrum Amsterdam (SARA). De rekentijd is beschikbaar gesteld door de stichting Nationale Supercomputer Facilitelen (NCF).
Contents

Summary ix
Samenvatting xi
Notation xiii

1 Introduction 1
1.1 Background of the research 1
1.2 Solution Method 2
1.3 Outline of this thesis 3

2 Governing Equations 5
2.1 Incompressible Magneto hydrodynamics 5
2.2 Scaling of the governing equations 7
2.3 The governing equations in general orthogonal curvilinear coordinates 10
2.4 Direct Numerical and Large-Eddy Simulations 11
2.4.1 The Large-Eddy equations 13
2.5 Review of Computational methods used in turbulence simulations 16
2.6 Finite Volume Discretization of an Orthogonal System 18

3 Straight pipe flow through an externally applied magnetic field 25
3.1 Introduction 25
3.2 Relation between the electric potential distribution and the fluid velocity 27
3.3 Experimental setup 29
3.4 Large Eddy Simulation 32
3.5 Results 36
3.5.1 Potential fluctuations and Spectra 39
3.6 Experimental results at high Reynolds numbers 43
3.7 Conclusion ............................................................................. 47

4 Turbulent flow in curved pipes .............................................. 49
  4.1 Introduction ........................................................................... 49
  4.2 Governing equations ............................................................. 51
  4.3 Results for Laminar flow ..................................................... 54
  4.4 Results for Turbulent flow .................................................. 56
    4.4.1 Computational details ..................................................... 57
    4.4.2 Mean flow quantities ...................................................... 59
    4.4.3 Secondary Motion .......................................................... 64
    4.4.4 Turbulent statistics ........................................................ 64
    4.4.5 Dissipation .................................................................... 72
    4.4.6 Production ..................................................................... 73
    4.4.7 Summary of the flow simulation ..................................... 74
  4.5 Response of an electromagnetic flowmeter on curved pipe flow ........................................................................ 75
  4.6 Conclusion ........................................................................... 78

5 Analogy between resistive MHD and viscous fluids ............ 81
  5.1 Introduction ........................................................................... 81
  5.2 The geometry ........................................................................ 82
  5.3 Results of the simulations .................................................... 86
  5.4 Relation with a resistive plasma .......................................... 92
  5.5 Conclusion ........................................................................... 97

6 Discussion and Conclusion .................................................. 99
  A Analytic Solution of the flow meter equation ......................... 103
    A.1 Some remarks on the 3D case .......................................... 106
  B Solution of Poisson’s equation ............................................ 107

Curriculum Vitae ......................................................................... 113
Electromagnetic effects in cylindrical pipe flow

Bendiks Jan Boersma

Summary

In this thesis the effect of turbulence and curvature is investigated on the reading of an electromagnetic flowmeter. In an electromagnetic flowmeter the flow of a (weakly) conducting fluid in a pipe is forced through a magnetic field. The electric potential which is induced according to the principle of Faraday is observed. This observation of the electric potential distribution can be used as a measure for the liquids bulk velocity. Flowmeters based on the principle sketched above have been used for several decades. Although the principle of such a flowmeter is reasonable well understood there remains details which needs further clarification. These are primarily related to turbulent flow phenomena.

In chapter one a short introduction to turbulence and magneto hydrodynamics is given. After this introduction, the important equations governing magneto hydrodynamic flows are given. These equations are first simplified for weakly conducting fluids. Next, a transformation of these equation to orthogonal curvilinear coordinates is given. Some possible discretization methods of these equations are discussed. One of these discretization methods (the so-called finite volume method) has been used to discretize these equations.

In chapter three the numerical model introduced in chapter two, is applied to calculate the electric potential distribution induced by a turbulent flow flowing through an external applied magnetic field. The turbulent flow is calculated with help of Large Eddy Simulation (LES) technique. The results of these computations
Summary

are compared with laboratory experiments. The agreement between the results obtained from experiments and simulations is in general good.

The by experiments validated numerical model is subsequently used to investigate a more complex flow, for instance the flow which occurs in a pipe bend. The numerical model is used to calculate the flow in such a geometry in detail. The results of these calculations have been compared with results given in the literature. Next, the calculated velocity fields are used to investigate the response of an electromagnetic flowmeter on such a flow. The difference in calculated electric potential in curved pipe flow and straight pipe flow is quite small, but non-negligible.

In chapter five an attempt has been made to use the mathematical analogy, which exist between the flow of an "ordinary" fluid and a resistive plasma, to investigate the transition to turbulence in such a plasma. First, the analogy between the two systems is described in detail. The presented results show that this analogy is not complete due to the presence of volume forces in the fluid problem.

In the last chapter the conclusions are given and discussed. The main conclusions of this thesis are that: the presented numerical techniques are capable to calculate the flow and the induced electric potential accurately. With help of the existing turbulence theory is possible to estimate the viscosity of the fluid, flowing through a flowmeter using the electric signal generated by the flowmeter. It has been shown that disturbances in the flow such as pipe bends have only a minor influence (approximately 5%) on the reading of a flowmeter. Furthermore, it is shown that the analogy which exists between viscous fluids and resistive plasmas fails at some points.
Electromagnetische effecten in cilindrische pijpstromingen

Bendiks Jan Boersma

Samenvatting

In dit proefschrift wordt onderzocht wat het effect van turbulentie in een stroming en kromming van de stroming is op het uitganssignaal van een elektromagnetische stromingsmeter. Een elektromagnetische stromingsmeter is gebaseerd op de wet van Faraday: De stroming van een (zwak) geleidende vloeistof in een pijp wordt door een magneetveld geleid. Volgens het principe van Faraday wordt een spanning geinduceerd die gerelateerd is aan de snelheid van de vloeistof. Stromingsmeters gebaseerd op het bovenstaande principe worden in de praktijk al zo'n dertig jaar gebruikt. Het principe van een stromingsmeter is redelijk eenvoudig, maar er zijn bepaalde details die niet geheel duidelijk zijn. Deze zijn hoofdzakelijk gerelateerd aan turbulentie.

Na een korte inleiding over turbulentie en magneto hydrodynamica in hoofdstuk een, worden in hoofdstuk twee de belangrijke vergelijkingen voor een magneto hydrodynamische stromingen gegeven. Deze algemene vergelijkingen kunnen vereenvoudigd worden voor zwak geleidende stromingen. Vervolgens worden enige transformatie regels gegeven die gebruikt kunnen worden om de vergelijkingen in orthogonale kromlijnige coördinaten te schrijven. Enkele mogelijke numerieke oplossingsmethoden voor deze vergelijkingen worden kort besproken. Hierna wordt een van deze methodes (de eindige volume methode) toegepast om de vergelijkingen te
discretiseren.

In hoofdstuk drie wordt de numerieke methode toegepast om de elektrische potentiële verdeling geïnduceerd door een turbulente pijlstroming stromend door een magneetveld te berekenen. De turbulentie in de stroming wordt gemodelleerd met behulp van de zogenaamde Large Eddy Simulatie (LES) techniek. De resultaten van het numerieke model worden vergeleken met de resultaten verkregen uit laboratorium experimenten. De overeenkomst tussen experiment en berekening is goed.

Nadat experimenteel is aangetoond dat het numerieke model goed werkt, wordt de uitbreiding gemaakt naar een meer complexe stroming zoals de stroming in een pijp bocht. Het numerieke model wordt gebruikt om de stroming in deze geometrie nauwkeurig uit te rekenen. De resultaten van deze berekeningen kunnen worden vergeleken met resultaten uit de literatuur. De berekende snelheidsvelden worden vervolgens gebruikt om de geïnduceerde elektrische spanning te berekenen. De elektrische potentiële verdeling die wordt berekend in een bocht wijkt niet veel af van de potentiële verdeling in een rechte buis. Het verschil zal voor metingen in technisch-commerciële toepassingen wel van belang zijn.

In hoofdstuk vijf wordt geprobeerd om m.b.v. de analogie die bestaat tussen stromingen van een "gewone" vloeistof en de stroming van een resistief plasma, iets te zeggen over het ontstaan van turbulentie in een zogenaamde tokamak. Eerst wordt uitgebreid ingegaan op de analogie. Uit de gepresenteerde resultaten blijkt duidelijk dat de analogie niet goed bruikbaar is om gedetailleerde voorspellingen te doen.

In het afsluitende hoofdstuk worden de conclusies getrokken. De belangrijkste hiervan zijn dat de gepresenteerde numerieke technieken in staat zijn om de stroming in een stromingsmeter nauwkeurig te berekenen. Met behulp van bestaande turbulentie theorie en metingen van de elektrische potentiële kan de viscositeit van de vloeistof worden geschat. De invloed van verstoringen zoals bochten op de uitvoer van een stromingsmeter is minstens een orde kleiner dan de invloed van deze verstoringen op de stroming zelf. Verder is het nog aangetoond dat de analogie tussen normale stromingen en stromingen van resistieve plasma's op bepaalde punten niet voldoende is om de turbulentie karakteristieken van de twee systemen vergelijkbaar te maken.
# Notation

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Meaning</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\mathbf{u}$</td>
<td>velocity vector</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$\rho$</td>
<td>mass density</td>
<td>$kg/m^3$</td>
</tr>
<tr>
<td>$P$</td>
<td>pressure</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$\tau$</td>
<td>stress tensor</td>
<td>$m^2/s^2$</td>
</tr>
<tr>
<td>$\nu$</td>
<td>kinetic viscosity</td>
<td>$m^2/s$</td>
</tr>
<tr>
<td>$\mathbf{J}$</td>
<td>electric current density</td>
<td>$A/m^2$</td>
</tr>
<tr>
<td>$\mathbf{B}$</td>
<td>magnetic induction</td>
<td>$T$</td>
</tr>
<tr>
<td>$\sigma$</td>
<td>electrical conductivity</td>
<td>$(\Omega m)^{-1}$</td>
</tr>
<tr>
<td>$\mathbf{E}$</td>
<td>electric field</td>
<td>$V/m$</td>
</tr>
<tr>
<td>$\mu$</td>
<td>magnetic permeability</td>
<td>$H/m$</td>
</tr>
<tr>
<td>$V$</td>
<td>electric potential</td>
<td>$V$</td>
</tr>
<tr>
<td>$\omega$</td>
<td>vorticity</td>
<td>$s^{-1}$</td>
</tr>
<tr>
<td>$C_f$</td>
<td>friction factor</td>
<td>$-$</td>
</tr>
<tr>
<td>$L$</td>
<td>length scale</td>
<td>$m$</td>
</tr>
<tr>
<td>$U$</td>
<td>velocity scale</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Kolmogorov length scale</td>
<td>$m$</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Kolmogorov time scale</td>
<td>$s$</td>
</tr>
<tr>
<td>$\epsilon$</td>
<td>dissipation rate</td>
<td>$m^2/s^3$</td>
</tr>
<tr>
<td>$\nu_t$</td>
<td>eddy viscosity</td>
<td>$m^2/s$</td>
</tr>
<tr>
<td>$C_s$</td>
<td>Smagorinsky constant</td>
<td>$-$</td>
</tr>
<tr>
<td>$\Delta_f$</td>
<td>large eddy filter</td>
<td>$m$</td>
</tr>
<tr>
<td>$L_{ij}$</td>
<td>Leonard stress</td>
<td>$Pa$</td>
</tr>
<tr>
<td>$S_{ij}$</td>
<td>strain-rate tensor</td>
<td>$s^{-1}$</td>
</tr>
<tr>
<td>$r$</td>
<td>radial coordinate</td>
<td>$m$</td>
</tr>
<tr>
<td>$f(r)$</td>
<td>dimensionless function</td>
<td>$-$</td>
</tr>
<tr>
<td>$u$</td>
<td>radial velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$v$</td>
<td>circumferential velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$w$</td>
<td>axial velocity</td>
<td>$m/s$</td>
</tr>
<tr>
<td>$\bar{U}$</td>
<td>bulk velocity</td>
<td>$m/s$</td>
</tr>
</tbody>
</table>
### Notation

<table>
<thead>
<tr>
<th>Term</th>
<th>Symbol</th>
<th>Unit</th>
</tr>
</thead>
<tbody>
<tr>
<td>pipe radius</td>
<td>$R$</td>
<td>m</td>
</tr>
<tr>
<td>pipe diameter</td>
<td>$D$</td>
<td>m</td>
</tr>
<tr>
<td>callibration constant</td>
<td>$G$</td>
<td>-</td>
</tr>
<tr>
<td>friction velocity</td>
<td>$u_*$</td>
<td>-</td>
</tr>
<tr>
<td>mixing length</td>
<td>$l_{mix}$</td>
<td>m</td>
</tr>
<tr>
<td>wave number</td>
<td>$k$</td>
<td>$m^{-1}$</td>
</tr>
<tr>
<td>autocorrelation</td>
<td>$\rho_{ii}$</td>
<td>-</td>
</tr>
<tr>
<td>streamfunction</td>
<td>$\psi$</td>
<td>$m^2/s$</td>
</tr>
<tr>
<td>radius of curvature</td>
<td>$R$</td>
<td>m</td>
</tr>
<tr>
<td>inverse radius of curvature</td>
<td>$\kappa$</td>
<td>$m^{-1}$</td>
</tr>
<tr>
<td>non conservative body force</td>
<td>$F$</td>
<td>N</td>
</tr>
<tr>
<td>ratio of polodial to toroidal velocity</td>
<td>$q$</td>
<td>-</td>
</tr>
</tbody>
</table>

### Dimensionless numbers

<table>
<thead>
<tr>
<th>Term</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reynolds number</td>
<td>$Re$</td>
</tr>
<tr>
<td>Alfven number</td>
<td>$Al$</td>
</tr>
<tr>
<td>magnetic Reynolds number</td>
<td>$Re_m$</td>
</tr>
<tr>
<td>Stuart number</td>
<td>$N$</td>
</tr>
<tr>
<td>Hartmann number</td>
<td>$Ha$</td>
</tr>
<tr>
<td>Dean number</td>
<td>$K$</td>
</tr>
</tbody>
</table>
Chapter 1

Introduction

1.1 Background of the research

Cylindrical pipes are frequently used for the transportation of liquids, examples of this are oil pipelines, water pipes which carry our drinking water and piping systems in industrial plants. In many cases information is needed about the flow rate through these pipes, for instance to detect leakage of the fluid from the pipeline or to measure the amount of fluid that is delivered to a customer. Depending on the desired accuracy needed to measure the flow rate one may chose a particular flow metering device. One of the most reliable flow metering devices for liquid pipe flows is probably the electromagnetic flowmeter. In such a flowmeter a magnetic field is imposed on the liquid transverse to the direction of the flow. The flowing liquid, which carries electrically charged ions, will generate an electric potential distribution according to Faraday’s law. This potential distribution, which is related to the vorticity in the flow, can be measured at the pipe wall. The magnitude of the potential tells something about the liquid velocity integrated over the pipe cross section and is thus related to the flow rate. An advantage of this flowmeter is the fact that the electric potential is in principle independent of the value of the electrical conductivity, which is related to the concentration of ions in the fluid. For a review of the existing theory on electromagnetic flow measurement we refer to Shercliff (1962), Bevir (1970) and Tsinober, Kit & Teitel (1987).

Although the principle of the electromagnetic flowmeter is reasonably simple and reasonably well understood there remain details which need further clarification. These are primarily related to the response of a flowmeter to turbulent flow and to non-ideal flow conditions, which are conditions that deviate from a fully developed pipe flow. Examples of this are the flow in a pipe bend, oscillating flows which can
occur after objects such as valves and swirling flows.

Here we will be mainly concerned with turbulent flow as this type of flow occurs in many practical applications. Turbulent flow occurs when the liquid velocity in a pipe reaches a certain critical value. In a turbulent flow several structures can be observed, which are usually called eddies. These eddies vary in size, from large to small. The size of the large eddies is related to the geometry of the flow, i.e. the pipe diameter. The size of the small eddies depends on the energy dissipation and viscosity. Each of these eddies rotates in the fluid and therefore carries its own vorticity. The electrical potential induced according to the principle of Faraday is dependent on the scalar product of the magnetic field and the vorticity (see e.g. Shercliff (1962)). So each eddy will have an influence on the induced electric potential in the flow. Therefore, detailed information on all the eddies in the flow is needed for a complete prediction of the electric potential.

In a fully developed cylindrical pipe flow the time averaged distribution of the vorticity will be independent of the circumferential direction and the orientation of the magnetic field is not important. However, for non-ideal flows such as the flow in a pipe bend, the vorticity distribution in the circumferential direction will be non-uniform due to the presence of a centrifugal force acting on the flow. This centrifugal force depends on the curvature of the pipe bend and the velocity in this bend. Because of the non-uniform distribution of the vorticity in this geometry also the orientation of the magnetic field becomes important.

The main goal of this thesis is twofold. First, we aim to investigate what precisely the effect is of turbulence on the reading of a flowmeter and second we aim to investigate the effect of non-ideal flow conditions, such as the flow in a pipe bend, on the reading of an electromagnetic flowmeter. In addition to these topics we will also investigate geometric resonances which can occur in fluid flow in a toroidal geometry. This is relevant for plasma physics, because there exist an analogy between fluid flow and the flow of a resistive plasma, which is used in the so-called plasma fusion reactors (tokamaks). Such a plasma can also be turbulent. With the model for the calculation of curved pipe flow we will investigate whether the turbulence in a plasma is caused by the same mechanism as turbulence in an ordinary fluid.

1.2 Solution Method

From the previous section it will be clear that for an accurate prediction of the induced electromagnetic potential detailed information on the vorticity in the fluid is needed. This quantity can not be measured easily in a laboratory experiment, because for an accurate measurement, the three velocity components at at least four points in the flow have to be measured simultaneously, which is very complicated
even with present days sophisticated experimental facilities. On the other hand it is quite straightforward to calculate the vorticity from a numerical simulation of the flow, because the three components of the velocity are known everywhere in the computational domain.

Therefore a numerical simulation will be very useful to predict the electric potential distribution in the flow. This requires an accurate simulation of the dynamics of the eddies, because this is essential for the calculation of the electrical potential distribution. Two well established techniques which are able to simulate the dynamics of these eddies are Direct Numerical Simulation (DNS) and Large Eddy Simulation (LES). Both techniques solve the three-dimensional time dependent Navier-Stokes equations, which are the equations which govern fluid motions. In a DNS all length and time scales of the flow are resolved on the computation grid. Due to computational constraints DNS is restricted to flows in simple geometries and to flows with a low Reynolds number, which is the non-dimensional number formed by the fluids velocity, length scale and viscosity. In a LES only the large scales (eddies) in a turbulent flow are resolved and the small eddies are modeled with a so-called subgrid closure model. This method can be used to calculate flows with higher Reynolds numbers, but the accuracy of the calculations will be less than in a DNS. In this thesis we will use both LES and DNS to investigate the response of an electromagnetic flowmeter on a turbulent flow. It is a priory not known if a LES contains enough information on the vorticity of the small scales needed for an accurate prediction of the electrical potential distribution. So a validation of the results by experimental data is needed. Therefore, we have also carried out an experiment of which the results also will be discussed.

1.3 Outline of this thesis

In this thesis we will use LES to investigate the effect of turbulence in a pipe on the reading of a flowmeter. Due to the uncertainties in the LES and also in the computation of the electrical potential distribution from the calculated flow field, an experimental validation of the results is valuable. To our knowledge their is no experimental data available on this flow. So we decided to build our own setup to validate the LES and the calculation of the electric potential.

DNS will be used to investigate the effect of curvature on the reading of a flowmeter. We have chosen for DNS because the reliability of it for complex flows is much higher than LES. We will also use DNS to investigate the analogy between the flow of resistive plasma in a tokamak geometry and the flow of a standard fluid in the same geometry.

The outline of this thesis is as follows: In chapter two we derive the governing
equations for the flow of a conducting liquid through an externally applied magnetic field. The resulting equations are simplified for a weakly conducting fluid like water. Based on these equation we calculate in chapter three the response of an electromagnetic flowmeter on straight turbulent pipe flow. The results of these calculations are validated by laboratory experiments. Next, in chapter four the by experiments validated numerical methods are applied to calculate the response of an electromagnetic flowmeter on the turbulent flow in a pipe bend. In chapter five we use the developed numerical model to study the analogy between viscous fluids and resistive plasmas. We end this thesis with our main conclusions and suggest some possible directions for future research in chapter six.
Chapter 2

Governing Equations

In this chapter we will present the governing equations for incompressible magneto hydrodynamic flow. We will non-dimensionalyse these equations and discuss the characteristic dimensionless parameters that appear in these equations. Next we point out how these equations can be written in an orthogonal curvilinear form. We will then discuss two solution techniques for the turbulent flow described by these equations, i.e. Direct Numerical and Large-Eddy Simulation. We will also shortly discuss some discretization methods for computing these solutions. Finally, we will consider a discretization of the governing equations by means of a Finite Volume technique.

2.1 Incompressible Magneto hydrodynamics

In this section we will first consider the equations which govern the flow of an incompressible fluid. Next, we will present the extensions which are necessary for a description of Magneto Hydrodynamic (MHD) flow.

The equations that describe the spatial and temporal evolution of a fluid flow have been known for a long time. These equations can be put in a form describing of conservation of mass, momentum and energy. Here we will only consider isothermal flows, so that we will not need the equation for the conservation of energy. For an incompressible fluid the conservation of mass is expressed in vector notation by the following relation, (Landau & Lifshitz 1989):

\[ \nabla \cdot \mathbf{u} = 0, \]

where \( \mathbf{u} \) is the velocity vector. The conservation of momentum reads, (Landau &
Table 2.1: Some typical values of the electrical conductivity (Moreau (1990))

<table>
<thead>
<tr>
<th>Material</th>
<th>$\sigma (\Omega^{-1} m^{-1})$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Distilled water</td>
<td>$\approx 10^{-4}$</td>
</tr>
<tr>
<td>Weak electrolytes (Sea water)</td>
<td>$10^{-4}$ to $10^{-2}$</td>
</tr>
<tr>
<td>Strong electrolytes (water + 25% NaCl)</td>
<td>21.6</td>
</tr>
<tr>
<td>Pure $H_2SO_4$</td>
<td>73.6</td>
</tr>
<tr>
<td>Liquid Mercury</td>
<td>$10^6$</td>
</tr>
</tbody>
</table>

Lifshitz 1989):

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} = -\frac{1}{\rho} \nabla P + \nabla \cdot \mathbf{\tau}. \quad (2.2)$$

in which $\rho$ is the mass density of the fluid, $P$ the pressure, and $\mathbf{\tau}$ the stress tensor. Equation (2.2) is also known as the Navier-Stokes equations. The stress tensor $\mathbf{\tau}$ is, for an incompressible Newtonian fluid, given by

$$\mathbf{\tau} = \nu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T). \quad (2.3)$$

In which $\nu$ is the kinematic viscosity of the liquid, and the superscript $T$ denotes the transpose of a vector. The equations (2.1)-(2.3) can describe all known flow phenomena of an incompressible, non-conducting Newtonian fluid.

Let us now consider the extension to a conducting fluid which flows through an externally applied magnetic field, $\mathbf{B}$. We assume that the fluid has an electrical conductivity $\sigma$ which is always positive. Some typical values of this electrical conductivity, for different fluids, are given in Table 2.1. The electrical conductivity of the fluid in combination with the magnetic field leads to two effects in the flow (e.g. Chandrasekhar (1961), Moreau (1990)): First, due to the motion of the electrically conducting fluid across the magnetic field lines, electric currents are generated and the secondary magnetic field associated with these electric currents contributes to changes in the primary or applied magnetic field. Second, the fact that fluid elements carrying currents traverse magnetic lines of force, leads to an additional force on the fluid elements. This force is better known as the Lorentz force.

To describe these effects resulting from the flow of a conducting fluid through a magnetic field the standard Navier-Stokes equations (2.2) must be extended by adding a volume force, i.e. the Lorentz Force and the set of equations must be enlarged by the Maxwell equations for the magnetic field.

---

1 Sometimes this force is called the Laplace force.
The Navier Stokes for the conducting fluid including the Lorentz force become:

\[
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla P + \nabla \cdot \mathbf{\tau} + \frac{1}{\rho} \mathbf{J} \times \mathbf{B}.
\]  

(2.4)

in which \( \mathbf{J} \) is the electric current density. The Maxwell equations which relate the electric and magnetic fields read (Moreau 1990):

\[
\nabla \cdot \mathbf{B} = 0,
\]  

(2.5)

\[
\nabla \times \mathbf{E} = -\frac{\partial \mathbf{B}}{\partial t},
\]  

(2.6)

\[
\nabla \times \mathbf{B} = \mu \mathbf{J}.
\]  

(2.7)

Here \( \mathbf{E} \) is the electric field and \( \mu \) the magnetic permeability. Furthermore there is a relation between the electric field and the current density, namely Ohm’s law:

\[
\mathbf{J} = \sigma \mathbf{E}.
\]  

(2.8)

In a moving fluid this equation remains correct for an observer moving with the local fluid velocity \( \mathbf{u} \). However in a fixed laboratory frame of reference this equation should be rewritten as

\[
\mathbf{J} = \sigma (\mathbf{E} + \mathbf{u} \times \mathbf{B}),
\]  

(2.9)

to which we will refer further on as Ohm’s law.

### 2.2 Scaling of the governing equations

We will now put the equations given in the previous section in a non-dimensional form. Let us assume that our problem can be characterised by a length scale \( L \), a velocity \( U_0 \) and magnetic-field induction \( B_0 \). Then, from equation (2.7) follows the quantity \( B_0/(L\mu) \) as a characteristic scale for the electric current. In the same way we get the characteristic quantity \( U_0 B_0 \) for the electrical field \( \mathbf{E} \). As time scale it is natural to use \( U_0/L \) and for the pressure \( \rho U_0^2 \). After dividing all terms by their appropriate scaling quantities, we obtain the following non-dimensional set of equations:

\[
\begin{align*}
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} &= -\nabla P + \frac{1}{Re} \nabla^2 \mathbf{u} + Al \mathbf{J} \times \mathbf{B}, \\
\nabla \times \mathbf{E} &= -\frac{\partial \mathbf{B}}{\partial t}, \\
\mathbf{J} &= \nabla \times \mathbf{B}, \\
\mathbf{J} &= Re_m (\mathbf{E} + \mathbf{u} \times \mathbf{B}).
\end{align*}
\]  

(2.10)
In which the following non-dimensional parameters appear. The \( Re = U_0 L/\nu \) is the standard Reynolds number, \( Re_m = \mu \sigma U_0 L \) the magnetic Reynolds number and \( Al = B_0^2/\mu \rho U_0^2 \) the Alfvén number. Let us first consider the influence of the magnetic Reynolds number, \( Re_m \). By combination of the equations (2.6), (2.7), and (2.9) we can obtain the following equation for the magnetic field induction
\[
\frac{\partial \mathbf{B}}{\partial t} = \nabla \times (\mathbf{u} \times \mathbf{B}) + \frac{1}{Re_m} \nabla^2 \mathbf{B}. \tag{2.11}
\]
So the time change of the induced magnetic field, \( \partial \mathbf{B}/\partial t \), is composed of a convective and diffusive term. If \( Re_m \gg 1 \) convection dominates over diffusion and if \( Re_m \ll 1 \) diffusion is dominant. The magnetic Reynolds number is in general very small for weakly conducting liquids. This can be seen from the following relation which can be derived between the Reynolds and magnetic Reynolds number
\[
Re_m = \nu \mu \sigma Re. \tag{2.12}
\]
Suppose we have a turbulent pipe flow of salt water with a Reynolds number of \( 10^6 \) and kinematic viscosity \( 10^{-6} m^2/s \), then the magnetic Reynolds number is
\[
Re_m = 10^{-6} \cdot 4\pi \cdot 10^{-7} \cdot 10^{-2} \cdot 10^{6} \approx 10^{-8}
\]
For such small values of the magnetic Reynolds number, the induced magnetic field will be negligible, i.e. \( \partial \mathbf{B}/\partial t \approx 0 \). Equation (2.6) then reduces to:
\[
\nabla \times \mathbf{E} = 0. \tag{2.13}
\]
This allows us to define an electric potential \( \nabla V = -\mathbf{E} \) and with this definition we can write Ohm's law, i.e. equation (2.9) as:
\[
\mathbf{J} = \sigma (-\nabla V + \mathbf{u} \times \mathbf{B}). \tag{2.14}
\]
Which, together with Kirchoff's first law (\( \nabla \cdot \mathbf{J} = 0 \)), leads to the following Poisson equation for the electric potential
\[
\nabla^2 V = \nabla \cdot (\mathbf{u} \times \mathbf{B}) = \mathbf{B} \cdot (\nabla \times \mathbf{u}) + \mathbf{u} \cdot (\nabla \times \mathbf{B}) = \mathbf{B} \cdot \omega. \tag{2.15}
\]
Here \( \omega = \nabla \times \mathbf{u} \) is the vorticity vector and we assumed that \( \nabla \times \mathbf{B} = \mu \mathbf{J} = 0 \). So for a medium with a small magnetic Reynolds number, the set of equations given by (2.10) reduces to
\[
\begin{align*}
\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla)\mathbf{u} &= -\nabla P + \frac{1}{Re} \nabla^2 \mathbf{u} + AlRe_m (-\nabla V + \mathbf{u} \times \mathbf{B}) \times \mathbf{B}, \\
\nabla^2 V &= \mathbf{B} \cdot \omega
\end{align*} \tag{2.16}
\]
In the following we will replace the advective term \((\mathbf{u} \cdot \nabla)\mathbf{u}\) by its conservative counterpart:
\[
(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot (\mathbf{uu}) + \mathbf{u}(\nabla \cdot \mathbf{u}) = \nabla \cdot (\mathbf{uu}),
\]
which is convenient for numerical calculations. The dimensionless group \(AlRe_m\) is also known as the interaction parameter \(N\) or Stuart number. The square root of the product of the interaction parameter and the Reynolds number gives the Hartmann number,
\[
Ha = \sqrt{NRe} = B_0L\sqrt{\frac{\sigma}{\rho \nu}},
\tag{2.17}
\]
The Hartmann number is an important number in magneto hydrodynamic flows because it relates the viscous forces to the Lorentz force. A flow with a large value of the Hartmann number will be influenced by the Lorentz force. For the parameters for the turbulent pipe flow, given previously, we obtain the following Hartmann number,
\[
Ha = B_0L\sqrt{\frac{10^{-2}}{1000 \cdot 10^{-6}}} = 3.1B_0L.
\]
Let us furthermore suppose that the pipe diameter is equal to 0.04\(m\) (which is the case for the experiments presented in chapter three) and the magnetic field \(B_0 = 1T\). Then the Hartmann number becomes 0.126. Branover (1978), gives the following empirical relation for the friction factor of a turbulent pipe flow flowing through a transversal magnetic field,
\[
\frac{C_f}{C_{f0}} = 1 - 1300 \left(\frac{Ha}{Re}\right)^{1.6}
\]
In which \(C_{f0}\) is the friction factor in ordinary pipe flow. For our numerical example for which \(Re = 10^6\) we obtain \(C_f/C_{f0} = 0.999997\). It will be clear that in this particular case the effect of the Lorentz force on the flow is negligible. To put the derived relations in another perspective we may give the following interpretation.

In applications where the fluid has a high electrical conductivity, like liquid metals or the flow of plasma in fusion reactors, the magnetic Reynolds number \(Re_m\) can be order one or more and the magnetic induction \(\mathbf{B}\) induced by the flow can be large. For applications with weakly conducting liquids like sea water or sulphuric acid the magnetic Reynolds number is much smaller than one and we can use the simplified set of equations (2.16). In which the magnetic induction \(\mathbf{B}\) is constant. If the Hartmann number \(Ha\) is also small then the Lorentz force in equation (2.16) is negligible and the flow is governed by the standard Navier-Stokes equation (2.2) and is decoupled from the equation for the electric potential (2.15).
2.3 The governing equations in general orthogonal curvilinear coordinates

In the previous section we have presented the governing equations in general vector notation. Here we will write down these equation in a orthogonal curvilinear form, which will be used for the calculations to be presented in the following chapters. In Figure 2.1 two systems of co-ordinates are sketched, a Cartesian system \((x, y, z)\) and a curvilinear system \((\eta_1, \eta_2, \eta_3)\) which are connected to each other by a coordinate transformation. Following Batchelor (1967) we can write the change in position vector \(\mathbf{x}=(x, y, z)\) as:

\[
\frac{d\mathbf{x}}{dx} = h_1 \frac{d\eta_1}{d\eta_1} \mathbf{e}_1 + h_2 \frac{d\eta_2}{d\eta_1} \mathbf{e}_2 + h_3 \frac{d\eta_3}{d\eta_1} \mathbf{e}_3, \tag{2.19}
\]

where \(\mathbf{e}_1, \mathbf{e}_2, \mathbf{e}_3\) are unit vectors in the curvilinear coordinate system, and \(h_1, h_2, h_3\) are the positive scale factors of the coordinate system. These scale factors can be obtained from the following relation:

\[
h_i = \sqrt{\left(\frac{\partial x}{\partial \eta_i}\right)^2 + \left(\frac{\partial y}{\partial \eta_i}\right)^2 + \left(\frac{\partial z}{\partial \eta_i}\right)^2}. \tag{2.19}
\]

With help of the the chain rule we can write

\[
\frac{\partial P}{\partial x} = \frac{\partial P}{\partial \eta_1} \frac{d\eta_1}{dx}
\]
Similar relations can be derived for \( \partial P/\partial y \) and \( \partial P/\partial z \). So the gradient operator \( \nabla \), in curvilinear coordinates can be written as

\[
\nabla P = \frac{1}{h_1} \frac{\partial P}{\partial \eta_1} e_1 + \frac{1}{h_2} \frac{\partial P}{\partial \eta_2} e_2 + \frac{1}{h_3} \frac{\partial P}{\partial \eta_3} e_3. \tag{2.20}
\]

For the divergence operator we find in a similar way, e.g. Morse & Feshbach (1953):

\[
\nabla \cdot \mathbf{u} = \lim_{dV \to 0} \frac{\int \mathbf{u} \, dA}{dV} = \frac{1}{h_1 h_2 h_3} \left( \frac{\partial}{\partial \eta_1} (h_2 h_3 u_1) + \frac{\partial}{\partial \eta_2} (h_1 h_3 u_2) + \frac{\partial}{\partial \eta_3} (h_1 h_2 u_3) \right), \tag{2.21}
\]

where \( dV = h_1 h_2 h_3 \, d\eta_1 \, d\eta_2 \, d\eta_3 \) is the volume of the elementary parallelepiped. For the curl of the vector \( \mathbf{u} \) we find:

\[
\nabla \times \mathbf{u} = \frac{e_1}{h_2 h_3} \left( \frac{\partial}{\partial \eta_2} (h_3 u_3) - \frac{\partial}{\partial \eta_3} (h_2 u_2) \right) + \frac{e_2}{h_1 h_3} \left( \frac{\partial}{\partial \eta_3} (h_1 u_1) - \frac{\partial}{\partial \eta_1} (h_3 u_3) \right) + \frac{e_3}{h_1 h_2} \left( \frac{\partial}{\partial \eta_1} (h_2 u_2) - \frac{\partial}{\partial \eta_2} (h_1 u_1) \right). \tag{2.22}
\]

Using the divergence, gradient, and curl operator in the form given above, we can write down the Navier-Stokes equations in an arbitrary orthogonal curvilinear system. Special care has to be taken into account to compute the term \( \nabla \cdot \mathbf{uu} \), because we must allow for the dependency of dyadic product \( \mathbf{u} \mathbf{u} \) and the unit vectors \( e_1, e_2, e_3 \), on the coordinates \( \eta_1, \eta_2, \eta_3 \). The result becomes

\[
\nabla \cdot (\mathbf{uu})_1 = \sum_n \frac{1}{h_n} u_1 u_n + \frac{u_2}{h_1 h_2} \left( \frac{\partial h_1}{\partial \eta_2} - \frac{\partial h_2}{\partial \eta_1} \right) + \frac{u_3}{h_1 h_3} \left( \frac{\partial h_1}{\partial \eta_3} - \frac{\partial h_3}{\partial \eta_1} \right). \tag{2.23}
\]

The other terms of \( \nabla \cdot \mathbf{uu} \), i.e. \( \nabla \cdot (\mathbf{uu})_{2,3} \) can be obtained by cyclic interchanges of the suffixes.

### 2.4 Direct Numerical and Large-Eddy Simulations

For small Reynolds numbers the viscous term \((\nu \nabla^2 \mathbf{u})\), in the Navier-Stokes equations dominates the convective term \((\nabla \cdot \mathbf{uu})\) and the flow will be laminar. For laminar flow the (numerical) solution of the Navier-Stokes equations is straightforward. For large Reynolds numbers the non-linear convective term in the Navier-Stokes equation \((\nabla \cdot (\mathbf{uu}))\) will dominate over the viscous term and, in general, the resulting flow will be turbulent. A turbulent flow is characterised by a broad range of length and time scales in the flow. As a result the numerical solution is complicated by the fact that all these length and time scales must be resolved. In the remainder of this section we will describe two popular techniques to solve the
Navier-Stokes equations for turbulent flow, i.e. Direct Numerical Simulation and Large-Eddy Simulation.

Direct Numerical Simulation (DNS) implies a fully resolved three-dimensional, time dependent numerical solution of the Navier-Stokes equations. In other words the evolution of all scales of motion are computed without use of any closure model or assumption. It is clear that DNS can only be used for those cases when the ranges of length scales in the flow becomes not too large. This will turn out to be a serious limitation to the application of DNS to simulate turbulence. Moreover, the large computing effort involved in DNS, has as consequence that DNS is mainly used to study turbulence physics, e.g. to help the development of turbulence models.

Let us now consider the range of length scales found in a turbulent flow in more detail. The smallest scales of motion in a turbulent flow are known as the Kolmogorov scales, which are the scales where turbulent energy is dissipated by viscosity (e.g. Tennekes & Lumley 1974). The Kolmogorov length and time scale are given by:

\[ \eta = \left( \frac{\nu^3}{\epsilon} \right)^{1/4}, \]  
\[ \tau = \left( \frac{\nu}{\epsilon} \right)^{1/2}. \]  

in which \( \nu \) is the kinematic viscosity of the fluid and \( \epsilon \) the turbulent dissipation in the flow. The dissipation rate \( \epsilon \), can be interpreted here as the rate of energy which is transported from the large scales to the small scales. Therefore the dissipation is limited by the energy flow from the large or macro scales of motion. Therefore \( \epsilon \) can be estimated from the following relation

\[ \epsilon = \frac{U^3}{L}, \]  

where \( U \) and \( L \) are the macroscopic velocity and length scale respectively. The relation given above can be interpreted as the ratio of the kinematic energy of the macro structure \( U^2 \) and time scale \( T = L/U \). Within a time scale, the eddy loses its kinetic energy due to break up in smaller eddies. These smaller eddies again break up into even smaller eddies. This process continues until the size of the eddies equals the Kolmogorov scales. This process is known as the “energy cascade”. In real turbulent flows there is indeed an energy transfer from the large to the small scales. However, transport from the small to the large scales is also possible. This transport from small to large scales is most times called “backscatter”. Hartel & Kleiser (1993) show that the energy transfer from the large to the small scales dominate over the energy transfer from small to large scales. However they also observe that backscatter is not negligible.
From combining equation (2.23) with (2.25) one can derive a relation for the ratio \( L/\eta \) as a function of the Reynolds number \( UL/\nu \). It then follows that an increasing Reynolds number leads to a larger \( L/\eta \) and in order to resolve all flow scales the computational grid and time must increase correspondingly. The following relation between the Reynolds number and the number of grid points \( N \) for a DNS, can be derived:

\[
N_{DNS}^3 \propto Re^{9/4}
\]

The memory of present day computers is limited to \( N^3 \approx 10^8 \). Therefore is will be clear that DNS is limited to flows with a low to a moderate Reynolds number.

A remedy to overcome the limitation of DNS for increasing \( Re \) is to artificially reduce the range of length and time scales that have to be computed. A possible method is Large Eddy Simulation (LES), in which the small scales are removed from the flow problem with help of a spatial filter. The remaining large scales can than be solved on a computational grid that requires less grid points than \( N_{DNS} \).

The small scales which are not resolved and which have been removed by the filter are called subgrid scales. Their influence on the resolved large scales can not be neglected and these scales need to be modelled with help of a so-called subgrid model. The choice of keeping only large scales is inspired by the fact that the large scale are mostly anisotropic and strongly depend on the geometry of a particular flow. The small scales are on the contrary mostly isotropic and do not depend strongly on the geometry of a particular flow. In other words the large eddies are characteristic for a specific turbulent flow. Moreover, the small scales contain much less energy than the large (resolved) scales. Therefore they will not have a large influence on the large eddies. The small scales are assumed to be more or less the same for every turbulent flow, and hopefully an universal sub-grid model for this scales can be developed.

To illustrate the main difference between DNS and LES we show in Figure 2.2 a typical three-dimensional energy spectrum of a turbulent flow (see Tennekes & Lumley 1974). In a DNS the whole area between the curve and x-axis is resolved. In a LES only the white region under the curve is resolved and the gray region is modelled with a subgrid model.

2.4.1 The Large-Eddy equations

The first step in a LES is the spatial filtering of the governing equations. A general filter operator can be defined according to

\[
\overline{f}(x, t) = \int \int \int_V G(x - x') f(x', t) dV
\]

(2.26)
where $G$ is the filter function, $x, x'$ are spatial vectors, and $f$ is a flow variable. After application of this filter operator to the Navier-Stokes equations, the filtered Navier-Stokes equations become:

$$
\frac{\partial \bar{u}}{\partial t} + \nabla \cdot (\bar{u} \bar{u}) = -\nabla \bar{p} + \nabla \cdot (\bar{\tau} + \tau_{sgs})
$$

(2.27)

The filtering introduces a new unknown, the so-called subgrid stress tensor $\tau_{sgs}$, which is equal to

$$
\tau_{sgs} = -(\bar{u}\bar{u} - \bar{u} \bar{u})
$$

(2.28)

This term represents that the subgrid scales, give rise to an additional stress acting on the resolved scales.

As $\tau_{sgs}$ is a new unknown we have more variables than equations and a closure model for $\tau_{sgs}$ is needed. Several models have been proposed to model this new unknown. Two popular models are the Smagorinsky model and the Dynamic Model\(^2\).

In the Smagorinsky model (see for instance Ghosal et al. (1995), Mason & Callen 1986) the unknown subgrid stress is parameterised as:

$$
\tau_{sgs} = \tau_{ij} = \nu_t \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right),
$$

(2.29)

\(^2\)This model is sometimes called the Germano model
where \( \nu_t \) is a turbulent or eddy viscosity. This viscosity is given by the following relation:

\[
\nu_t = (C_s \Delta_f)^2 \sqrt{2 S_{ij} S_{ij}}, \quad S_{ij} = \frac{1}{2} \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right).
\]  

(2.30)

where \( \Delta_f \) is the filter width and \( C_s \) the so-called Smagorinsky constant. Its value can be related to the energy spectrum in the inertial subrange. In practice this constant varies between 0.1 and 0.25 (Mason & Callen (1986)). For shear driven flows a typical value of the Smagorinsky constant of 0.1 is used and for the simulation of homogeneous turbulence approximately 0.2. The main disadvantage of the Smagorinsky model is that the physical process which it represents is too simple for the dynamics of the subgrid scales. In principle the Smagorinsky model drains energy from the large scales to the small scales. However, on the subgrid level we know that the energy transport can go both ways from large to small but also from small to large. This latter process is known as backscatter. Only on the average the energy transfer must go form large to small scales. The Smagorinsky model is thus not capable to model backscatter in the flow. Furthermore the Smagorinsky model can not be used for transitional calculations because it lacks the property to turn off when the flow is laminar, i.e. whenever there is a velocity gradient the Smagorinsky model produces a stress and thus an eddy viscosity.

To improve the Smagorinsky model Mason & Thomson (1992) have included in it a small amount of random backscatter, resulting in somewhat better results for their atmospheric boundary layer calculations. However in their model a second adjustable constant appeared, which determines the amount of backscatter.

Another subgrid model has been proposed by Germano et al. (1991), the so-called Dynamic model. In this model a second or test filter is introduced, besides the LES filter. The width of the test filter is larger than the width of the (LES) grid filter. In the dynamic model the (double) filtered subgrid stress is given by

\[
T_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j,
\]

where a tilde denotes the test filter. By applying the test filter also to the subgrid stress \( \tau_{sgs} \), given by (2.28) the following result can be derived:

\[
L_{ij} = \bar{u}_i \bar{u}_j - \bar{u}_i \bar{u}_j.
\]

\( T_{ij} \) and \( L_{ij} \) are related to each other by the following relation

\[
L_{ij} = T_{ij} - \tilde{\tau}_{sgs},
\]

which is known as the Germano identity. The resolved turbulent stress \( L_{ij} \) can be computed by applying the test filter to the LES results. Furthermore it is assumed
that the same subgrid model can be used for $T_{ij}$ and $\tau_{sgs}$. For the case of the Smagorinsky model we obtain, after some algebra, the following equation

$$L_{ij} \overline{S_{ij}} = -2C_s (\overline{\Delta^2} - \Delta_J^2)|\overline{S}||\overline{S_{ij}} S_{ij}$$

(2.31)

which only contains resolved scales. From equation (2.31) $C_s(x, y, z, t)$ can be obtained, in a least square sense. In principle this model does not have an adjustable constant anymore and therefore it is preferable above the Smagorinsky model. Furthermore the model can give a negative value of $C_s$ which could be interpreted as backscatter.

However, the dynamic model has also some drawbacks. First, locally in the flow the coefficient for $C_s$ in the right hand side term of equation (2.31) can become zero resulting in a very large value of the $C_s$. Second, because $C_s$ can be negative, the sum of the molecular and turbulent viscosity ($\nu + \nu_t$) can be smaller than zero, which is physically and numerically not acceptable. To circumvent the problems mentioned above, most times the constant $C_s$ is averaged in the homogeneous directions and/or in time.

Recently a new approach to subgrid modelling has been developed by Sullivan et al. (1995) who introduce two different computational grids. In general the mesh size is related to the size of the eddies in the flow. So a fine mesh is used in regions with small eddies and coarse meshes in regions with large eddies. The relation between the subgrid stresses on both meshes is again given by $L_{ij}$. The coarse grid covers the whole computational domain and the fine grid covers only a part of the coarse grid. On the fine grid and the coarse grid outside the region of the fine grid a simple subgrid model like the Smagorinsky model is used. In the overlap region the fine grid simulation is used to calculate the subgrid stress tensor $\tau_{sgs}$ of the coarse grid. With this model it is possible to increase the accuracy of the coarse grid simulation in regions where this is necessary, for instance near the wall or in a region with large velocity gradients (see for instance Boersma et al. (1996)). Furthermore this model can give backscatter in the nested grid regions without the complications caused by dynamic model. The disadvantage of this model is that there is still a Smagorinsky constant on the fine grid which has to be set. However its expected that the coarse grid simulation is not very sensitive to the value of the Smagorinsky constant on the fine grid, because the fine grid is much better resolved than the coarse grid.

### 2.5 Review of Computational methods used in turbulence simulations

In this section we will shortly review some of the existing solution methods for the equations given in section 2.2. These are: spectral methods, finite element methods
and finite volume/difference methods.

**Spectral methods**

Spectral methods use a finite number of orthogonal basis functions in which the velocity and pressure are expanded. The most frequent used basis functions are trigonometric polynomials, Chebyshev polynomials and Legendre Polynomials. Trigonometric polynomials are used in combination with periodic boundary conditions. Chebyshev and Legendre polynomials are used in combination with Dirichlet or Neumann boundary conditions. Due to the choice of these special basis functions there is a severe limitations on the geometry of the flow problem. The main advantage of spectral methods is that they have the highest possible numerical accuracy for a given number of grid points. Spectral methods are therefore preferable for problems which require a high numerical accuracy on all flow scales, like transitional calculations. For a detailed review on spectral methods we refer to Canuto *et al.* (1988) and Gottlieb & Orszag (1977)

**Finite Element Method**

In a Finite Element Method the computational domain is subdivided into simple elements of arbitrary shape and size. Within each element a certain number of nodal points are defined. These nodal points are the points where the numerical values of the unknown functions and their derivatives have to be evaluated. The field variables are then approximated by means of polynomials which are locally defined within each element, and set to zero outside the element. The polynomials are integrated over each element, using a Gauss quadrature method with the nodal points as basis points. After this integration a matrix vector equations for the unknown velocity components and pressure remains, which has to be solved. For three dimensional problems the matrix becomes extremely large. The solution of this matrix vector system can be rather time consuming and requires a large amount of core memory. The main advantage of a FEM above other methods is that it is very suited to tread complex geometries. For a review on finite elements methods we refer to Zienkiewicz & Taylor (1989).

**Finite Volume Method**

In a Finite Volume Method the governing equations are integrated over a small grid volume. The volume integrals can be evaluated as surface integrals using Gauss theorem. The surface integrals are evaluated numerically, for instance with the midpoint rule. The resulting discretization is very similar to the discretization
following from a standard finite difference method. The main advantage of the finite volume method above a finite difference method is the fact that it can be used on arbitrary orthogonal meshes. For a review on this method we refer to Hirsch (1988).

For stationary problems all three methods mentioned above, require approximately the same amount of core memory. For time dependent problems, where explicit time stepping is used, Finite Volume and Spectral methods require less memory than Finite Element methods, because in the latter two methods it is not necessary to store a large matrix.

The core memory needed for the FEM, FVM and SM can be estimated from

\[
M_{FEM} \propto N_1 \times N_2 \times N_3 \times N_{\text{band}}, \\
M_{FVM} \propto N_1 \times N_2 \times N_3, \\
M_{SM} \propto N_1 \times N_2 \times N_3.
\]

(2.32)

In which \( N_i \) is the number of grid points in the \( i^{th} \) coordinate direction and \( N_{\text{band}} \) the band width of the FEM matrix which is in general:

\[
N_{\text{band}} = \mathcal{O}(N_i^2).
\]

So for a grid with \( N_1 = N_2 = N_3 = N \) we need for a FVM or SM \( \mathcal{O}(N^3) \) bytes and for a FEM method \( \mathcal{O}(N^5) \). It will be clear that for large 3D applications (\( N > 100 \)) only Finite Volume and Spectral Methods can be used. In this thesis we will use a Finite Volume method because it is more flexible, with respect to the boundary conditions than a spectral method.

### 2.6 Finite Volume Discretization of an Orthogonal System

In this section we will give a more detailed description of the application of a Finite Volume discretization to an orthogonal coordinate system. A so-called staggered grid is used for the discretization of the continuity and Navier-Stokes equations, i.e. the pressure and velocity points are located at different points of the grid volume. The advantage of a staggered grid above a collocated grid is that it does not suffer from the checker-board splitting of the pressure that collocated (non-staggered) grid can have (see Hirsch (1988)).

In Figure 2.3 we have sketched a staggered grid cell with the pressure point at the centre of the cell and the three velocity points at the cell sides.
Figure 2.3: A grid cell of the staggered grid, \(dx = h_1 d\eta_1, dy = h_2 d\eta_2\), and \(dz = h_3 d\eta_3\).

Discretization of the governing equations

The continuity equation in an orthogonal coordinate system reads:

\[
\frac{1}{h_1 h_2 h_3} \left( \frac{\partial}{\partial \eta_1} [h_2 h_3 u_1] + \frac{\partial}{\partial \eta_2} [h_1 h_3 u_2] + \frac{\partial}{\partial \eta_3} [h_1 h_2 u_3] \right) = 0. \tag{2.33}
\]

We will integrate this equation around the pressure point (see Figure 2.3):

\[
\int_{\eta_1 - \Delta \eta_1/2}^{\eta_1 + \Delta \eta_1/2} \int_{\eta_2 - \Delta \eta_2/2}^{\eta_2 + \Delta \eta_2/2} \int_{\eta_3 - \Delta \eta_3/2}^{\eta_3 + \Delta \eta_3/2} \frac{1}{h_1 h_2 h_3} \left( \frac{\partial}{\partial \eta_1} [h_2 h_3 u_1] + \frac{\partial}{\partial \eta_2} [h_1 h_3 u_2] + \frac{\partial}{\partial \eta_3} [h_1 h_2 u_3] \right) h_1 h_2 h_3 d\eta_1 d\eta_2 d\eta_3, \tag{2.34}
\]

or

\[
\int_{\eta_1 - \Delta \eta_1/2}^{\eta_1 + \Delta \eta_1/2} \int_{\eta_2 - \Delta \eta_2/2}^{\eta_2 + \Delta \eta_2/2} \int_{\eta_3 - \Delta \eta_3/2}^{\eta_3 + \Delta \eta_3/2} \left( \frac{\partial}{\partial \eta_1} [h_2 h_3 u_1] + \frac{\partial}{\partial \eta_2} [h_1 h_3 u_2] + \frac{\partial}{\partial \eta_3} [h_1 h_2 u_3] \right) d\eta_1 d\eta_2 d\eta_3. \tag{2.35}
\]

This volume integral can be reduces to three surface integrals, of which we will only give the first one

\[
\int_{\eta_2 - \Delta \eta_2/2}^{\eta_2 + \Delta \eta_2/2} \int_{\eta_3 - \Delta \eta_3/2}^{\eta_3 + \Delta \eta_3/2} [h_2 h_3 u_1]_{\eta_1 - \Delta \eta_1/2} d\eta_2 d\eta_3. \tag{2.36}
\]
Using the midpoint rule for the numerical integration we can write:

\[
\Delta \eta_2 \Delta \eta_3 \left[ h_2 h_3 u_1 \big|_{(\eta_1 + \Delta \eta_1/2, \eta_2, \eta_3)} - h_2 h_3 u_1 \big|_{(\eta_1 - \Delta \eta_1/2, \eta_2, \eta_3)} \right] + O[(\Delta \eta_2)^2, (\Delta \eta_3)^2],
\]

(2.37)

The whole discretization of the continuity equation then becomes:

\[
\nabla \cdot \mathbf{u} = \\
\Delta \eta_2 \Delta \eta_3 \left[ h_2 h_3 u_1 \big|_{(\eta_1 + \Delta \eta_1/2, \eta_2, \eta_3)} - h_2 h_3 u_1 \big|_{(\eta_1 - \Delta \eta_1/2, \eta_2, \eta_3)} \right] + \\
\Delta \eta_1 \Delta \eta_3 \left[ h_1 h_3 u_2 \big|_{(\eta_1, \eta_2 + \Delta \eta_2/2, \eta_3)} - h_1 h_3 u_2 \big|_{(\eta_1, \eta_2 - \Delta \eta_2/2, \eta_3)} \right] + \\
\Delta \eta_1 \Delta \eta_2 \left[ h_1 h_2 u_3 \big|_{(\eta_1, \eta_2, \eta_3 + \Delta \eta_3/2)} - h_1 h_2 u_3 \big|_{(\eta_1, \eta_2, \eta_3 - \Delta \eta_3/2)} \right]
\]

which is second-order accurate in space.

Similar discrete relations can be derived for the Navier-Stokes equations. As an example we will give the discretization of the second convective term in the Navier-Stokes equation in the \( \eta_1 \) direction, which reads

\[
\frac{1}{h_1 h_2 h_3} \frac{\partial}{\partial \eta_2} (h_1 h_3 u_1 u_2)
\]

(2.38)

This equation is integrated round the \( u_1 \) velocity point,

\[
\int_{\eta_1}^{\eta_1 + \Delta \eta_1} \int_{\eta_2 - \Delta \eta_2/2}^{\eta_2 + \Delta \eta_2/2} \int_{\eta_3 - \Delta \eta_3/2}^{\eta_3 + \Delta \eta_3/2} \frac{1}{h_1 h_2 h_3} \\
\frac{\partial}{\partial \eta_2} (h_1 h_3 u_1 u_2) \, h_1 h_2 h_3 \, d\eta_1 \, d\eta_2 \, d\eta_3,
\]

(2.39)

Partial integration gives

\[
\int_{\eta_1}^{\eta_1 + \Delta \eta_1} \int_{\eta_3 - \Delta \eta_3/2}^{\eta_3 + \Delta \eta_3/2} \left[ h_1 h_3 u_1 u_2 \right|_{\eta_2 - \Delta \eta_2/2}^{\eta_2 + \Delta \eta_2/2} \, d\eta_2 \, d\eta_3
\]

with help of the midpoint rule we can write this as.

\[
\Delta \eta_1 \Delta \eta_3 \left[ h_1 h_3 \left( \frac{u_1|_{\eta_1 + \frac{1}{2} \Delta \eta_1, \eta_2, \eta_3} + u_1|_{\eta_1 - \frac{1}{2} \Delta \eta_1, \eta_2, \eta_3}}{2} \right) - \\
\left( \frac{u_2|_{\eta_1, \eta_2 + \frac{1}{2} \Delta \eta_2, \eta_3} + u_2|_{\eta_1, \eta_2 - \frac{1}{2} \Delta \eta_2, \eta_3}}{2} \right) \right] \\
- h_1 h_3 \left( \frac{u_1|_{\eta_1 + \frac{1}{2} \Delta \eta_1, \eta_2, \eta_3} + u_1|_{\eta_1 + \frac{1}{2} \Delta \eta_1, \eta_2 - \Delta \eta_2, \eta_3}}{2} \right) \\
+ \left( \frac{u_2|_{\eta_1, \eta_2 - \frac{1}{2} \Delta \eta_2, \eta_3} + u_2|_{\eta_1 + \Delta \eta_1, \eta_2 - \frac{1}{2} \Delta \eta_2, \eta_3}}{2} \right) \right]
\]

(2.40)

\[+ O[(\Delta \eta_1)^2, (\Delta \eta_3)^2].\]
Similar discretization for the convective and diffusive terms can be derived in the other coordinate directions.

**Boundary conditions**

In this section we will describe the different types of boundary conditions which can be used in combination with the spatial discretization given above. We distinguish the following boundary conditions: periodic conditions, boundary conditions for the velocity and boundary conditions for the stress.

When the flow on two opposite sides of the computational domain is the same, for instance the tangential direction in a pipe, periodic boundary conditions are required, i.e. the velocity and pressure at the two sides of the computational domain are exactly the same. On solid walls, in general, no-slip boundary conditions have to be used, i.e. the velocity must be prescribed. The pressure must be left unspecified at a solid wall, because otherwise the problem would be over determined. On "open" boundaries, in general, a boundary condition for the stress $-P\delta_{ij} + \nu(\nabla u + (\nabla u)^T)$ can be used.

**Temporal Discretization**

In the previous section we have given the spatial discretization of the Navies-Stokes equations. In this section we will point out how the integration in time will be carried out. The time derivatives in the Navier-Stokes equations:

$$\frac{\partial u_i}{\partial t} = f_i(\eta_1, \eta_2, \eta_3), \quad i = 1..3,$$

are replaced by finite difference approximations, viz.

$$\frac{\partial u}{\partial t} = \frac{1}{\Delta t}(au^{n+1} + bu^n + cu^{n-1} + ..).$$

In which the superscript $n$ denotes a discrete time level, $\Delta t$ the time step, and $a, b, c$ are constants. The function $f_i(\eta_1, \eta_2, \eta_3)$, which stands for the spatial discretization, can be evaluated at different time levels. So the general form of a discrete time integration scheme becomes:

$$\frac{1}{\Delta t}(au^{n+1} + bu^n + ..) = Af_i^{n+1}(\eta_1, \eta_2, \eta_3) + Bf_i^n(\eta_1, \eta_2, \eta_3) + ..$$

When the coefficient $A$ is zero the scheme is explicit and when $A \neq 0$ the scheme is called implicit. The implementation of an explicit scheme is in general straightforward, while the implementation of an implicit scheme can be very complicated. Furthermore, an explicit time step is in general (computationally) much cheaper
than an implicit step. However, explicit schemes have also a limitation on the time step, due to stability requirements which most implicit schemes do not have.

For simulations of turbulent flow the time step is also limited by the physical process. For instance, for a DNS the time step should be smaller than the advection time scale. Another example is the vortex shedding behind an object. In this case the time step should be much smaller than the characteristic shedding time. For turbulent flow this physical allowed time step is of the same order of the time step for an explicit scheme. Therefore, for most time dependent turbulent flow calculations explicit time integration schemes have to be used. In the following we discuss the two time integration methods which we have used in this thesis.

**Method 1: Leap-Frog/Euler Forward.**

In this method, which was introduced by Schumann (1973), the advection is treated with a Leap-Frog scheme and diffusion with an Euler-Forward scheme. Eggels et al. (1994) used this scheme for the radial and axial direction in their pipe flow calculations. In the circumferential direction they replaced the explicit schemes with implicit schemes, i.e. Crank-Nicolson for the advection and Euler-Backward for the diffusion. With this partly implicit treatment the time step is not limited by the small grid cells near the pipe centreline. This implicit/explicit scheme reads

\[
\frac{u^{n+1} - u^{n-1}}{2\Delta t} = -A^n_{expt} - \frac{1}{2} A^{n-1}_{impl} - \frac{1}{2} A^{n+1}_{impl} + D^n_{expt} + D^{n+1}_{impl},
\]  

(2.41)

where \(A\) stands for the advection and \(D\) for the diffusion. For stability reasons the Euler-forward/backward scheme is taken over \(2\Delta t\). The computational costs for this partly implicit scheme (one direction) is comparable to a fully explicit scheme, because the matrices which occur due to the implicit treatment are tri-diagonal. For such systems very efficient solution methods are available, Golub & van Loan (1996).

The velocity \(u_+^{n+1}\) will in general not satisfy the continuity equation, because continuity is enforced by the pressure and this quantity has no time derivative and is thus not known at the new time level. Here we will use the so-called pressure correction method (van Kan (1986), Chorin (1968)) to calculate the pressure at time level \(n + 1\) and to correct the velocity in such a way that it satisfies the continuity equation. The pressure correction equation reads:

\[
\frac{u^{n+1} - u_+^{n+1}}{2\Delta t} = -\nabla P^{n+1}.
\]

(2.42)

\(^3\)A weak filter is used to remove the computational mode of the Leap-Frog scheme (see Pourquie (1994)).
Taking the divergence this equation gives
\[ -\nabla \cdot \mathbf{u}^{n+1} = -2\Delta t \nabla^2 P^{n+1}, \] (2.43)
in which we have forced the divergence at time level \( n + 1 \) to be zero. The Poisson equation (2.43) has to be solved every time step (see appendix A). The boundary conditions for the pressure \( P^{n+1} \) are chosen in such a way that they do not conflict with the prescribed boundary conditions for the velocity.

The maximum allowed time step for this scheme is given by Wesseling (1996),
\[ \Delta t \leq \frac{\beta}{\left| \frac{u}{\Delta \eta_1} \right| + \left| \frac{u}{\Delta \eta_2} \right| + 4\nu \left[ \left( \frac{1}{\Delta \eta_1} \right)^2 + \left( \frac{1}{\Delta \eta_2} \right)^2 \right]}, \] (2.44)
in which \( \beta \) is a safety factor smaller than one, and in which we have assumed that the second coordinate direction is treated implicitly. This time integration scheme is formally only first order accurate in time, with an error of \( O(2\Delta t) \).

**Method 2: Adams-Bashforth**

The second time integration scheme is based on Adams-Bashforth schemes (see for instance Gear (1971)). The second-order accurate explicit Adams-Bashforth scheme \( (a = 1, b = 1) \) reads
\[ \frac{\mathbf{u}_*^{n+1} - \mathbf{u}^n}{\Delta t} = \frac{3}{2}(-A^n + D^n) - \frac{1}{2}(-A^{n-1} + D^{n-1}). \] (2.45)
The pressure correction equation for this scheme reads:
\[ \frac{\mathbf{u}^{n+1} - \mathbf{u}_*^{n+1}}{\Delta t} = -\nabla P^{n+1}, \] (2.46)
which is very similar to the equation (2.42) given above. The maximum allowed time step for this scheme is given by the following heuristic criterium\(^4\):
\[ \Delta t \leq \frac{\beta}{\left| \frac{u}{\Delta \eta_1} \right| + \left| \frac{u}{\Delta \eta_2} \right| + \left| \frac{w}{\Delta \eta_3} \right| + \nu \left[ \left( \frac{1}{\Delta \eta_1} \right)^2 + \left( \frac{1}{\Delta \eta_2} \right)^2 + \left( \frac{1}{\Delta \eta_3} \right)^2 \right]} \] (2.47)
The error of this time integration scheme is \( O(\Delta t^2) \).

---

\(^4\) A better but more complicated criteria for the maximum time step of Adams-Bashforth scheme is given by Wesseling (1996)
For applications where the time step limitation is large due to small grid spacings in one direction, for instance near the centre of a pipe, we will use the Leap-Frog/Euler forward method (chapter 3 and 4). For other applications where the time limitation for an explicit scheme is not too large we prefer the more accurate Adams-Bashforth method (chapter 5).

An important step in the discretization scheme is the solution of the Poisson equation for the pressure, e.g. equation (2.42),(2.46). It is essential that this equation is solved with sufficient accuracy, to obtain an acceptable divergence of the velocity field. Several methods for the solution of Poisson’s equation have been developed in the past. These method can be roughly subdivided in two categories, namely direct and iterative techniques. In general iterative techniques are faster and require less memory than direct methods. However, for special cases where the Poisson equation is separable in one or more directions, the use of direct solution method can be very efficient, both in usage of computing time and memory. In appendix B such a direct solution technique is described for a Poisson equation which is separable in at least one of the coordinate directions.
Chapter 3

Straight pipe flow through an externally applied magnetic field

In this chapter we will both theoretically and experimentally investigate the response of an idealised electro magnetic flow meter on turbulent pipe flow. With help of large eddy simulation we calculate the flow field and the electric potential distribution induced by the magnetic field. In the experiment we measure the electric potential distribution at the pipe wall. The agreement between the measured and computed electric potential at the pipe wall is in general very good. From these results it follows that LES is a very useful tool to study or optimise electromagnetic flowmetering devices.

3.1 Introduction

In the previous chapter we have derived the equations (2.16) which govern the motion of a weakly conducting fluid through an externally applied magnetic field. From these equations it follows that a fluid with a non-zero electrical conductivity which flows through an externally applied magnetic field, generates an electric potential distribution. This electric potential distribution is in principle independent of the conductivity, density and the viscosity of the fluid, and depends only on the externally applied magnetic field strength, a length scale, and the fluid vorticity. When the magnetic field is known, there is a direct relation between the vorticity, i.e. the rotation of the velocity, and the induced electric potential in the fluid. In
this chapter we will consider the fundamentals of the electromagnetic flowmeter and its operation. Based on the equations, that are derived, we will implement these in a numerical simulation model for turbulent pipe flow. The resulting computations are then compared with the experiment that we have carried out in this project. Before, we discuss the details of our numerical simulations and the experiments, we first consider the principle of the electromagnetic flowmeter.

The principle of electromagnetic flowmetering is based upon a law discovered by Faraday. This law gives the relation between electric potential and magnetic induction. Faraday also was one of the first researchers who used this observation to measure the flow rate of a liquid. In an experiment in 1832 he tried to measure the flow rate in the Thames river by measuring the potential difference, induced by the earth magnetic field, between two pillars of the Waterloo Bridge. Faraday’s experiment failed, probably because the river bed short-circuited the electric signal. Since this early work of Faraday, many oceanographers have studied the sea currents, waves and tides successfully with his method. Williams (1930) was probably the first who used the induced electric potential to measure the flow rate in a pipe. In Figure 3.1 we have sketched the basic geometry of Williams his flow meter, which is still the principle used in commercial available flow meters.

The principle consists of conducting fluid which is flowing through an electrical isolated circular pipe. Perpendicular to the main flow direction, a (homogeneous) magnetic field $B$ is applied. The induced electric potential is measured at the pipe wall where two electrodes are placed perpendicular to the magnetic field. The
induced electric potential is given by the following relation (e.g. equation (2.15)):

\[ \nabla^2 V = B \cdot \nabla \times u, \quad J(R) = \sigma \frac{\partial V}{\partial r} \bigg|_{r=R} = 0, \quad (3.1) \]

where \( V \) is the electric potential. The electric current is taken to be zero at the pipe wall, because the pipe is electrically isolated and the fluids velocity is zero at the wall. For normal operating conditions the magnitude of the electric potential is rather small, \( \mathcal{O}(10^{-3} V) \), and amplification of the signal is thus necessary in order to make it observable.

The layout of this chapter is as follows: In section two we will derive the relation between the electric potential distribution, the pipe diameter and the fluid velocity. In section three we will describe the experiments and in section four the numerical simulation. Next, in section five and six we will give some results from the experiment and simulations. Finally, in section seven a conclusion will be given.

3.2 Relation between the electric potential distribution and the fluid velocity

In this section we will derive a relation between the induced electric potential and the fluid velocity for a the case of a homogeneous magnetic field which is applied perpendicular to the pipe axis. In view of the cylindrical pipe geometry we will use cylindrical coordinates, i.e. \( \eta_1 = r, \eta_2 = \theta, \eta_3 = z \) with \( r, \theta, z \) the radial, tangential and axial coordinate respectively. The scale factors for these coordinates are

\[ h_1 = 1, \quad h_2 = r, \quad h_3 = 1. \]

When we assume that the magnetic field is two-dimensional and homogeneous, i.e. \( B = (B \cos \theta, B \sin \theta, 0) \) (see Figure 3.1) we can rewrite equation (3.1) as:

\[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial V}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 V}{\partial \theta^2} + \frac{\partial^2 V}{\partial z^2} = B \cos \theta \left( \frac{1}{r} \frac{\partial w}{\partial \theta} - \frac{\partial v}{\partial z} \right) + B \sin \theta \left( \frac{\partial v}{\partial z} - \frac{\partial w}{\partial r} \right), \quad (3.2) \]

where \( u, v \) and \( w \) denote the velocity components in the radial, circumferential and axial directions respectively. If we furthermore assume that the flow is fully developed and axis-symmetric, it follows that

\[ \frac{\partial w}{\partial r} \gg \frac{1}{r} \frac{\partial w}{\partial \theta}, \quad \frac{\partial}{\partial r} \gg \frac{\partial}{\partial z}, \]
Equation (3.2) then simplifies to:

$$\frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial V}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 V}{\partial \theta^2} = -B \sin \theta \frac{\partial w}{\partial r}.$$  

This differential equation can be solved by substituting $V(r, \theta) = f(r) \sin \theta$, which gives the following ordinary differential equation for $f(r)$:

$$r^2 \frac{d^2 f}{dr^2} + r \frac{df}{dr} - f = \frac{d}{dr} \left( r^2 \frac{df}{dr} - rf \right) = -Br^2 \frac{\partial w}{\partial r}$$

integrating this equation from the pipe centre $r = 0$, to the pipe wall $r = R$, gives

$$R^2 \frac{df}{dr}(R) - Rf(R) = - \int_0^R r^2 B \frac{\partial w}{\partial r} dr = -r^2 B w|_0^R + B \int_0^R 2rw dr = R^2 B \overline{U}.$$  

where $\overline{U}$ is the mean velocity given by

$$\overline{U} = \frac{1}{\pi R^2} \int_0^{2\pi} \int_0^R w(r, \theta) rdrd\theta = \frac{2}{R^2} \int_0^R w(r)rdr. \quad (3.3)$$

In view of our assumption that the pipe wall is isolated and the velocity at the wall is zero it follows that $df/dr(R) = 0$. So we find for $V(R, \theta)$ the following relationship

$$V(R, \theta) = -B\overline{U}R \sin \theta. \quad (3.4)$$

From this relation it follows that the electric potential at the pipe wall is only a function of the mean velocity, the pipe radius, and the magnetic field strength and is independent of the velocity gradients in the fluid. The potential difference between the electrodes in Figure 3.1 is equal to $2RB\overline{U}$. So the bulk velocity can be calculated from the following relation:

$$\overline{U} = \frac{\Delta V_{max}}{2RB} = \frac{\Delta V_{max}}{B \overline{D}} \quad (3.5)$$

where $\Delta V_{max}$ is the maximum potential difference over the perimeter of the pipe.

In practical situations where the pipe diameters can be large, it is difficult and expensive to generate a homogeneous magnetic field over the whole pipe section. Therefore, in commercial flowmeters most times a inhomogeneous magnetic field is used. The solution of the flow meter equation (3.2) becomes in this case much more complicated. In this general case the potential difference between the two electrodes can be expressed as (Shercliff (1962)):

$$\Delta V = GBD\overline{U},$$
where \( G \) is in principle a function of \( B \) and \( u \), i.e. of both the value and direction of the magnetic and velocity field. However, for a proper chosen magnetic field, the influence of \( u \) and \( B \) on \( G \) is small. Therefore \( G \) is assumed to be constant and its value is obtained by calibration against a known bulk velocity.

In the sequel of this chapter we will restrict ourselves to a homogeneous magnetic field, i.e. \( G \) is a constant and thus independent of the Reynolds number of the flow.

### 3.3 Experimental setup

In this section we will shortly describe the setup in which our experiments have been carried out and the measurement procedures that we have used to obtain the experimental data. In Figure 3.2 we show a sketch of the experimental facility. The fluid is flowing from the settling chamber into a straight pipe, which is made of perspex. The pipe has a length of approximately eight meters and an internal diameter of \( 4.0 \cdot 10^{-2} \text{m} \). As a general rule a turbulent pipe flow is considered to be fully developed after approximately 50 pipe diameters. So the flow can be considered as fully developed at approximately two meters downstream of the settling chamber. The pressure gradient in the pipe is measured with two pressure tabs. The distance between the pressure tabs is four meters. The first pressure tab is located three meters downstream of the settling chamber, i.e. in the fully developed region. The measurement section with the homogeneous magnetic field is located six meters downstream of the settling chamber.
The magnetic field is generated by two large coils placed in a iron yoke (see Figure 3.3). The magnet is designed is such a way that the magnetic field is almost homogeneous (inhomogeneity smaller than 1%) over the pipe diameter (see Verheul (1994)). Two power supplies are used to generate an electric current of maximal 200 amperes. However, most of the experiments are carried out with a current of 100 ampere (to avoid overheating of the coils) resulting in a magnetic field strength of about $0.46T$ (esla) and a heat production of approximately 1000 Watt. The magnetic field generated by this magnet is much stronger than the field used in commercial flow meters. In the latter the magnetic field is typically of order 0.05 $T$. We have chosen for such a strong magnetic field because the signal to noise ratio of the electric potential increases with increasing strength of magnetic field. An important noise source is, for instance, the electro chemical effect at the interface between liquid and electrode (see Shercliff (1962), Tsinober, Kit & Teitel (1987) ). On the other hand the disadvantage of such a strong magnet is that the coils must be cooled in some way, because of the large heat production.

![Figure 3.3: A sketch of the magnet.](image)

In the wall of the perspex pipe four pairs of (round) stainless steel electrodes, with a diameter of 2mm are placed (see Figure 3.4). Ten centimetre down stream of the electrodes a stainless steel ring is placed along the perimeter of the pipe, which is electrically connected to the earth. Each pair of electrodes which are positioned diametrically opposite from each other are connected to four instrumentation amplifiers which amplify the signal between two electrodes and the stainless steel ring 103.2 times. The amplified signal is measured with an AD converter connected to a personal computer on which the measurement signal is stored. After each experiment the data are transferred to a HP720-workstation for data analysis.

Downstream of the measurement section a commercial electro magnetic flow
Figure 3.4: A sketch of the measurement section, with eight electrodes and magnetic field.

The meter is located which can be used independently to measure the flow rate in the pipe. As another check on the flow rate measurement we have also weighted the mass of the water flowing through the pipe over a period of 60 seconds.

As an example we show in Figure 3.5 a typical signal measured between the electrodes 1&5 which make an angle of $\pm \pi/2$ with the magnetic field. The Reynolds number in this measurement is approximately 20,000 and the magnetic field $\approx 0.7T$. The magnetic field which is gradually switched on after 6 seconds, yields a potential difference over these electrodes of approximately 20mV. The signal remains constant over a period of at least 30 seconds. When the measurement is continued for a longer period, say $t > 1000 \text{sec}$, the potential difference between the electrodes will decrease due to electro chemical effects which result in pollution of the electrodes. In addition we show in Figure 3.6 the power spectrum of the last twenty seconds of the signal plotted in Figure 3.5. From this power spectrum it is clear that there are preferred frequencies present in the signal. These are caused by the electric mains ($n \cdot 50Hz, \quad n = 1,2,3..$) and these have to be removed from the signal to allow for a proper physical interpretation. This is done using Fourier filtering in the following way: An at equal time interval sampled time series is transformed to the Fourier space with help of a Fast Fourier Transform (FFT), Brigham (1975). The value of the Fourier series at frequencies in between $n49Hz$ and $n51Hz$ are linear interpolated, where $n$ takes the values 1, 2, and 3. Furthermore the values of all Fourier components above a frequency of $200Hz$ are set to zero. The modified Fourier series is then transformed back to the physical space with the inverse of the FFT. This results in a time series from which the influence of the interference signal
at 50Hz has been removed. The error made by the interpolation of the frequencies is small (≈ 1%) because most energy is located in the low frequencies (< 10Hz), (see e.g. Figure 3.6) and these frequencies are not altered by the filtering procedure.

Due to the electrochemical pollution of the interface between the stainless steel electrode and the fluid it is possible that there exists an additional electric potential difference between two electrodes. This potential difference can also be present when the fluid is completely at rest. Its magnitude can be $O(0.1V)$, which is comparable or larger than the potential difference induced by the magnetic field. This potential depends on, several parameters such as for instance, the temperature and the salinity of the fluid. These parameter are difficult to control. So it is necessary to eliminate this potential from the measurement. This can be done in the following way; We start the measurement of the electric potential without a magnetic field. After a certain time (approximately 5sec) the magnetic field is switched on and the electric potential is induced according to the principle of Faraday (see Figure 3.5). The measurement is continued for a time period of 35s. The average electric potential over the first 5 seconds is computed and this value is subtracted from the signal, i.e.

$$\Delta V_{0.40sec} := \Delta V_{0.40sec} - \bar{V}_{0.5sec}.$$  

### 3.4 Large Eddy Simulation

In this section we will shortly describe the large-eddy simulation of turbulent pipe flow and the computation of the potential distribution, due to an externally imposed
Figure 3.6: Power spectra of the signal shown in Figure 3.5, with $20 < t < 40s$. Top: normal scales, Bottom: logarithmic scales. The Reynolds number is approximately 20,000 and the magnetic field $0.7T$. 
uniform magnetic field.

The LES model which we have used has been developed by Eggels (1994). The main assumption is that the flow in the pipe is fully developed, i.e. the statistics are independent of the axial coordinate. This fact allows us to use periodic conditions between the out and inflow cross section. Which is very convenient, because a correct formulation is far from trivial. The length of the periodic domain, used in simulations is equal to five pipe diameters, which is sufficient to obtain reliable statistics, (see Eggels et. al. (1994), Eggels (1994)). The flow in the pipe is driven by a constant mean pressure gradient in the axial direction. The value of this pressure gradient, can be found from a simple force balance in the axial direction which (under the assumption that the flow is fully developed) gives:

$$\nabla P = \frac{\Delta P}{\Delta z} = -\rho \frac{4u_*^2}{D}. \quad (3.6)$$

Here \(u_*\) is the friction velocity which by definition is equal to \(\sqrt{\tau/\rho}\) with \(\tau\) as wall shear stress.

The subgrid model used in the simulations is a simple Smagorinsky model (see Chapter 2). A so-called Van Driest damping function (see e.g. Piomelli et al. 1988):

$$l_{mix} := l_{mix}[1 - \exp(y^+/26)], \quad \text{with} \quad y^+ = \left[\frac{D}{2} - r\right] \frac{u_*}{v}, \quad (3.7)$$

is used to reduce the mixing length of the Smagorinsky model in the near wall region. With this modification the turbulent eddy viscosity used in the Smagorinsky model becomes zero at the wall, which should be the case because of the fact that the Reynolds stresses are zero there.

The numerical grid used in the calculations consists of \(36 \times 128 \times 192\) grid points in the radial, circumferential, and axial direction respectively. The grid in the radial direction is non-uniform, so it is possible to resolve the near wall layer, where large gradients are present. In addition to the LES for the velocity field we must in this case also calculate the induced electric potential. For this we need according to equation (2.15) the vorticity \(\omega\). This quantity has been calculated at the edge of the grid cells (Figure 2.3) by using Stokes law. The vorticity is interpolated to the centre of the grid cell. Next the right hand side of the flow meter equation is calculated, using the two dimensional homogeneous magnetic field, given previously, and \(\omega\). The solution of the resulting Poisson equation (3.2) yields the electric potential distribution in the whole computational domain.

The large eddy simulations have been started from a theoretical velocity profile (Hinze (1975)) for a fully developed turbulent pipe flow, upon which small random fluctuations are superimposed. The calculations are run for a sufficiently long time (approximately ten time scales \(D/u_*\)) in order for the flow to be independent of
Table 3.1: Some quantities used in or obtained from the simulations. $N_r, N_\theta, N_z$ denotes the number of grid points in the three coordinate directions, $Re_*$ the Reynolds number based on the friction velocity, $Re_b$ the Reynolds number based on the bulk (mean) velocity, $C_S$ the Smagorinsky constant, and $B$ the magnetic field.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>$N_r$</td>
<td>36</td>
</tr>
<tr>
<td>$N_\theta$</td>
<td>128</td>
</tr>
<tr>
<td>$N_z$</td>
<td>192</td>
</tr>
<tr>
<td>$Re_*$</td>
<td>$1.54 \cdot 10^3$</td>
</tr>
<tr>
<td>$Re_b$</td>
<td>$29.0 \cdot 10^3$</td>
</tr>
<tr>
<td>$C_S$</td>
<td>0.1</td>
</tr>
<tr>
<td>$B$</td>
<td>1</td>
</tr>
<tr>
<td>$\overline{U}$</td>
<td>19.8</td>
</tr>
</tbody>
</table>

the initial conditions. After this initial computation period the calculations have been continued for another five time scales during which fifty data fields are stored, for data analysis. In addition to these stored data fields we also record the electric potential at certain positions in the pipe as a function of time. The LES calculations are performed on the CRAY-C90 of the SARA-computing centre (Amsterdam) and took approximately 40 CPU hours.

The experiments are characterised by a Reynolds number based on the bulk velocity. This dimensionless number is defined as

$$Re_b = \frac{\overline{U}D}{\nu}.$$  \hspace{1cm} (3.8)

In the simulation of pipe flow we fix the pressure gradient and not the flow rate through the pipe, so the bulk Reynolds number is not known apriori. The pressure gradient which corresponds to a certain bulk Reynolds number can be estimated from the well known Blasius correlation:

$$C_f = 0.0791 Re_b^{-1/4}, \quad u_* = \frac{1}{2} \sqrt{2C_f \cdot \overline{U}},$$

and equation (3.6). The simulations are started with this pressure gradient. Then the mean velocity is calculated from the simulations and $u_*$ is adjusted. With this iteration method we obtain a value of 1540 for the Reynolds number based on the frictional velocity. (see also Table 3.1).
3.5 Results

In this section we will present some results obtained from the LES and compare them with experimental data.

Mean profiles

In Figure 3.7 we show the mean velocity profile obtained from the simulations carried out at a Reynolds number of $Re_b = 29 \cdot 10^3$. In the same figure we also give the "law of the wall" velocity profiles (e.g. Hinze (1975)) which read:

$$ W = \frac{u^2 y}{\nu} \quad y u_*/\nu < 11, \quad \text{with} \quad y = D/2 - r $$

$$ W = 2.5 \log \left( \frac{yu_*}{\nu} \right) + 5.5 \quad y u_*/\nu > 30, \quad (3.9) $$

and the experimental results of Toonder (1996) obtained at a Reynolds number of $24.8 \cdot 10^3$. The agreement between simulation, approximation and measurements is very good in the viscous sublayer, i.e. $y^+ = u_* y / \nu < 5 \quad (y = D/2 - r)$ which is not surprisingly because in this layer the boundary conditions, have a large influence. Outside this layer the agreement between simulation, logarithmic approximation and measurement is less good which is probably caused by the incorrect subgrid modelling near the wall. Eggels (1994) has argued that this yields a too high bulk velocity, which is also visible in Figure 3.7. In the following we will show that this deviation in the simulated velocity profile has only a minor influence on the results for the electric potential.

In Figure 3.8 we show the mean induced electric potential at the pipe wall obtained from the LES, as a function of the circumferential angle $\theta$. In this figure we have also plotted the exact solution of the flowmeter equation for an axisymmetric velocity profile, given by (3.4), where $\bar{U}$ is calculated from the LES data using the midpoint rule. The agreement between the analytical and numerical solution is excellent. The maximum potential difference $\Delta V_{max}$ is within 0.2 % as obtained from equation (3.4) by using the calculated bulk velocity. As the solution (3.4) is an exact result which directly follows from the flowmeter equation, it seems that our numerical solution of the flow meter equation (3.1) gives acceptable results.

Turning to the experiment we show in Figure 3.9 the electric potential difference between the electrodes that are diametrically placed opposite to each other (after normalising but before filtering) as a function of time. The maximum potential difference occurs between electrodes 1-5 and is equal to 0.013 Volts. This potential should be related to the bulk velocity, e.g. equation (3.4). From the experiment the bulk velocity can be calculated by weighting the mass flowing through the pipe
Figure 3.7: The mean axial velocity profiles as a function of the non-dimensional distance to the wall. The boxes denote the results of the Large Eddy Simulation, the circles the experimental results of Toonder (1996) and the lines the law of the wall, e.g. equation (3.9).

Figure 3.8: The mean electric potential at the pipe wall. The line denotes the exact solution and the symbols are obtained with the LES model.
Figure 3.9: The measured electrode potential between electrodes. The magnetic field is switched on at t = 4 sec. Top: The electric potential between electrodes 1-5 and 3-7. Bottom: The electric potential between electrodes 2-6 and 4-8.
over a period of 60 seconds. The bulk velocity then follows from:

$$\bar{U} = \frac{1}{4\rho \pi D^2} \frac{Q}{\bar{Q}}$$

where $Q$ is the mass-flow per second. The mass-flow in the experiment was found to be equal to 0.89 kg/sec which yields a bulk velocity of 0.708 m/s. The flow rate has also been measured with the commercial flow meter (Krohne - Altimeter). This flow meter yields almost the same bulk velocity (difference smaller than 1%). Therefore we conclude that in the present experiment the bulk velocity is equal to 0.71 m/s with an error of 1%.

Based on the measured $\bar{U}$ we can compute with help of (3.4) the $\Delta V_{\text{max}}$. By substituting in (3.4) we find $2\bar{U}RB = 2 \cdot 0.708 \cdot 0.02 \cdot 0.46 = 0.013 \text{Volts}$. This is within measuring accuracy equal to the measured potential difference given above. In other words our flowmeter has a calibration constant $G$ which is equal to unity, i.e. the flow meter behaves exactly as expected.

In Figure 3.9 (bottom) we have plotted the electrode potential at the electrodes which make an angle of $+45^\circ$ and $-45^\circ$ with the magnetic field. The absolute value of the potential at these electrodes should be equal to $\Delta V_{\text{max}} \cos(45^\circ) = 0.09 \text{Volts}$, and this agrees with the measurements. In Figure 3.9 (top) we also show the potential at the electrodes parallel to the magnetic field. The potential difference between these electrodes is negligible, which is consistent with the theory.

From the results shown in this section we can conclude that the mean quantities obtained from the experiments are in very good agreement with the results obtained from the LES. Furthermore, both are equal to the analytic solution of the flow meter given in section 3.2 for the case of a homogeneous magnetic field.

### 3.5.1 Potential fluctuations and Spectra

In this section we will compare the fluctuations in the electric potential observed in the experiment with the fluctuations obtained from the LES.

The fluctuations in the measured signals shown in Figure 3.9 are almost the same with and without magnetic field. The explanation is that these fluctuations are mainly caused by the electric mains as we have argued in section 3.5. In Figure 3.10 we show again the measured electrode potential, but now without the contribution of the electric mains, which has been obtained by using the correction procedure discussed in section 3.3. The figure shows clearly that the fluctuations in the electric potential become larger when a magnetic field ($t > 10\text{sec}$) is turned on. The fluctuations in the potential are dependent on the value of the magnetic induction $B$. To study these fluctuations in some detail we introduce the following
quantities

\[ u_{rms} = \sqrt{u'^2}, \quad v_{rms} = \sqrt{v'^2}, \]

\[ w_{rms} = \sqrt{w'^2}, \quad V_{rms} = \sqrt{V'^2}, \quad \hat{V}(\omega) = \sum_{0}^{n} V(t)e^{-i\omega t} \]

where \( u', v', w' \) are the velocity fluctuations in the radial, tangential, and axial direction respectively, and \( V' \) the fluctuation in the electric potential. Note that each fluctuation has been defined with respect to an appropriate mean, i.e. \( \overline{u'} = \overline{v'} = \overline{w'} = \overline{V'} = 0 \).

In Figure 3.11 we show the computed profiles of \( u_{rms}, v_{rms}, w_{rms} \) as a function of \( r/D \) in combination with the experimental data obtained by Perry & Abell (1975) at a Reynolds of 80,000 (all the profiles are scaled with \( u_* \)). The agreement between experiment and simulation is acceptable when we take the difference in Reynolds number between experiment and simulation into account and also the possible error in the LES due to the subgrid model as discussed in section 3.4.

In Figure 3.12 we have plotted the simulation results for the root mean square values of the electric potential, \( V_{rms} \), scaled with \( BD\overline{U} \), at the pipe wall as a function of the circumferential angle. The figure shows that there is no clear dependence on the circumferential angle, which is in agreement with the equations given in the appendix A. Therefore we will use in the following the average value of \( V_{rms} \), over the circumferential direction, \( \rightarrow V_{rms} = 0.095BD\overline{U} \).

In Table 3.2 we have listed the root-mean-square values calculated from the signals plotted in Figures 3.9 after filtering (i.e. according to Figure 3.10). The root mean square values are almost the same at all four electrode pairs. In Table 3.2 we have also listed the root-mean-square value obtained from the LES, i.e.
\[ \sqrt{2} \cdot \text{V}_{\text{rms}} \]

The agreement between computed and measured root-mean-square value is reasonable and seems to indicate the correctness of the LES computations. Hence the small difference which we have found for the mean profiles, as shown in Figure 3.7, does not seem to have a large influence on the computed electric potential.

It is clear that the fluctuations in the velocity are much larger than the fluctuations in the electric potential. As an example we have plotted in Figure 3.13 the electric potential obtained from the LES at \( y^+ = 7 \) (near electrode 1) as a function of time. In this figure we have also included the axial velocity, scaled with \( u_* \), as a function of time at the same distance of the wall. We see from this figure again that fluctuations in the electric potential are much smaller than the fluctuations in the velocity field. The explanation of this observation is simple. The fluctuations in the electric potential, at the pipe wall, are related to the integral of the velocity fluctuations over the cross section of the pipe (Bevir (1970)). So most of the fluctuations in the velocity field are integrated out which results in a reasonable smooth electric potential field.

In Figure 3.14 (top) we have plotted the power spectra \( \hat{V}^2 \) of the measured potential without a magnetic field. In Figure 3.14 (bottom) we have plotted the same spectra with the magnetic field switched on. The difference between both Figures is large for the low frequencies and negligible for the high frequencies, i.e. only the low frequencies are amplified by the magnetic field, which is in agreement with the theoretical results given in appendix A. Both spectra exhibit clearly the frequencies of the electric mains (50 Hz, 100Hz...). In Figure 3.15 we have plotted again the power spectra given in Figure 3.14 (bottom) but now in combination with the time spectra obtained from the time series of the electric potential which has been recorded during the LES at the same circumferential positions as in the experiment. In both cases the spectra have been scaled with \( (\vec{BDU})^2 \). The agreement between the measured and computed spectra is quite good considering all numerical and experimental uncertainties. This figure also confirms that fluctuations in the electric potential are independent of the circumferential angle \( \theta \).

Unfortunately, with our simple measurement technique we are not able to measure spatial spectra. However it is possible to get some insight in the spatial structures with help of the Taylor hypothesis. Using the Taylor hypothesis we can write down the following simple dispersion relation:

\[ k = \overline{U} \omega. \]

\(^1\)The variance between electrodes a and b is equal to

\[ \text{var}(a + b) = \text{var}(a) + \text{var}(b) \]

if \( \text{var}(a) = \text{var}(b) \) we can write \( \text{var}(a + b) = 2\text{var}(a) \) and \( \text{rms}(a + b) = \sqrt{2}\text{rms}(a) \)
Figure 3.11: The root mean square profiles of the velocity components obtained from the LES (lines), and the experimental data of Perry and Abdel (symbols). The solid line denotes the radial rms, the dashed line the tangential rms, and the dotted line the axial rms.

Figure 3.12: The computed root mean square values at the pipe wall as a function of the circumferential angle (scaled with $DB\bar{U}$).
Figure 3.13: Upper line: The axial velocity at \( y^+ = 7 \) as a function of time (obtained from the LES). Lower line: The induced electric potential at \( y^+ = 7 \) as a function of time.

In which \( \overline{U} \) is the bulk velocity. So the time spectrum can be transformed into a one-dimensional spatial spectrum.

In appendix A we show that if the flow has an energy spectrum with a \( k^{-5/3} \) slope, i.e. the inertial subrange (Tennekes & Lumley (1974)), the electric potential should have spectrum with a \( k^{(-5/3) - 2} \) slope. In Figure 3.15 we have plotted a line with such a slope and we indeed see that this line forms a good fit to the data for a range of frequencies. From this observation we can draw the following conclusion: The fluctuations in the electric potential are mainly generated by the very large scales in the flow, thus scales which have a typical length scale of a tenth of the pipe diameter or more. Hence LES is ideally suited for the computation of the electrical potential field.

### 3.6 Experimental results at high Reynolds numbers

In this section we will present results of experiments carried out at higher Reynolds numbers. We will do this for the following Reynolds numbers 53000, 83000, 130000. We will only present results for the potential from the electrode pair which makes an angle of 90 degrees with \( \mathbf{B} \), because in the previous section we have shown that the effects are the same on each electrode pair (e.g Figure 3.14).

To start we show in Figure 3.16 the time spectra of the electric potential for
Figure 3.14: Top: The power spectra of the measured potential at the diametrically placed electrodes, with $B = 0T$. Bottom: The power spectra of the measured potential, with $B = 0.46T$. 
Figure 3.15: The lines denote the power spectra obtained from the measurements (see also Figure 3.14). The symbols denote the power spectra obtained from the LES. Furthermore the solid line has a slope of $-11/3$.

the three bulk Reynolds numbers, given above. All the spectra are normalised with $(BDU)^2$. From Figure 3.16 it follows that the "energy" in the small frequencies, i.e. the large scales according to the Taylor hypothesis, decreases with increasing Reynolds number. At the same time the higher frequencies, i.e. the smaller scales become more important. For large frequencies we expect that the spectrum behaves as (appendix A)

$$\delta^2 = C \varepsilon^{2/3} k^{-11/3}.$$  \hspace{1cm} (3.11)

where $C$ is a constant, $\varepsilon$ the dissipation rate and $k$ a wave number. The mean dissipation rate in a pipe flow is proportional to the work done by the pressure gradient, thus

$$\varepsilon \propto \frac{\partial P}{\partial z} \bar{U}.$$  

For the pressure gradient we can write

$$\frac{\partial P}{\partial z} \propto u_z^2 = C_f \bar{U}^2,$$

where $C_f$ is the friction coefficient. If the Reynolds number is large we can use the Blasius correlation for $C_f$, which reads $C_f = 0.0791/(\bar{U}D/\nu)^{1/4}$. It then follows

$$\varepsilon \propto \frac{\partial P}{\partial z} \bar{U} \propto \bar{U}^2 \bar{U}^{-1/4} \bar{U} = \bar{U}^{11/4}.$$
Table 3.2: Some quantities used and obtained from the experiments

<table>
<thead>
<tr>
<th>Electrodes</th>
<th>angle $\theta$</th>
<th>$V_{mean}(V)$</th>
<th>$\sqrt{V_t^2/V_{max}}$ (exp)</th>
<th>$\sqrt{V_t^2/V_{max}}$ (LES)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1-5</td>
<td>$90^\circ$</td>
<td>$-1.3 \cdot 10^{-2}$</td>
<td>$7.7 \cdot 10^{-3}$</td>
<td>$6.9 \cdot 10^{-3}$</td>
</tr>
<tr>
<td>2-6</td>
<td>$135^\circ$</td>
<td>$-9.1 \cdot 10^{-3}$</td>
<td>$6.2 \cdot 10^{-3}$</td>
<td>$6.9 \cdot 10^{-3}$</td>
</tr>
<tr>
<td>3-7</td>
<td>$180^\circ$</td>
<td>$-2.2 \cdot 10^{-6}$</td>
<td>$7.0 \cdot 10^{-3}$</td>
<td>$6.9 \cdot 10^{-3}$</td>
</tr>
<tr>
<td>4-8</td>
<td>$225^\circ$</td>
<td>$-9.3 \cdot 10^{-3}$</td>
<td>$7.5 \cdot 10^{-3}$</td>
<td>$6.9 \cdot 10^{-3}$</td>
</tr>
</tbody>
</table>

Because our experiments are carried out with a constant viscosity and pipe diameter we can replace the mean velocity by the Reynolds number, so that $\epsilon \propto Re^{11/4}$.

In view of the scaling sketched above we expect that the spectra collapse when Figure 3.16 is scaled with $[Re^{11/4}]^{2/3} = Re^{11/6}$. In Figure 3.17 we show the spectra scaled in this way and it is clear that a the high frequencies the spectra collapse reasonable well on to a single curve! This implies that the spectrum for the electric potential, at high frequencies, scales with $\epsilon^{2/3}$, just like the velocity in the fluid.

The constant in equation 3.11 is independent of the flow conditions, and likely depend only of the magnetic field. Once this constant is known for a flowmeter, equation 3.11 can be used to estimate the dissipation in the fluid. From the dissipation rate and the known liquid velocity and pipe diameter it is possible to obtain the Reynolds number of the flow, i.e. the viscosity of the fluid can be estimated.

Finally we show Figure 3.18 the autocorrelation, $\rho(\tau)$ of the potential measured between electrodes 3-7 for the three different Reynolds numbers. The integral time scale defined as

$$T = \int_0^\infty \rho(\tau)d\tau$$

(3.12)

decreases with increasing Reynolds number. This time scale is related to the time scales of the macro scales in the electric potential and thus to the macro scales of the velocity. Consequently for $\tau \approx T$ the correlation describes the macro scales in the flow and for $\tau \approx 0$ the micro scales in the flow. Both time scales decrease with increasing Reynolds number.

The results presented in Figures 3.16 - 3.18 in combination with the equations presented in appendix A, show that it is possible to predict the spectra and correlations of the electric potential from the spectra and correlations of the fluid. This is very useful because much information on velocity correlations and spectra in fluid
Figure 3.16: Power spectra of the electric potential between electrode 1 and 5 at different Reynolds numbers.

is known, while this is not the case for the potential. Another application is that by measuring the electric potential, which is less complicated than measuring the velocity, one can get insight in the time and length scales of the flow.

3.7 Conclusion

In this section we have investigated the response of an electromagnetic flow meter for a fully developed pipe flow both numerically and experimentally. For one typical case we have shown that the agreement between experiment and computation is excellent for the mean flow quantities. The agreement for second order statistics is good. It is shown that the fluctuations in the electric potential are at least one order smaller than the fluctuations in the velocity field, which is caused by the elliptic nature of the electric field in its dependence on the velocity fluctuations. Due to this elliptic nature of the flow meter equation only very large length and time scales in the flow contribute to the electric potential generation. Therefore Large Eddy Simulation (LES), which resolves the large flow scales, is a very useful tool to investigate the response of such a flowmeter. Furthermore, it is shown that theory which is available for the spectra of the velocity components and energy in fluids can be used to get insight in the spectra of the electric potential.
Figure 3.17: Power spectra of the electric potential between electrode 1 and 5 at different Reynolds numbers, scaled with $Re^{11/6}$.

Figure 3.18: Autocorrelation of the signal measured between electrode 1 and 5 at different Reynolds numbers.
Chapter 4

Turbulent flow in curved pipes

The mean flow and turbulent statistics obtained from the direct numerical simulation of the fully developed turbulent flow through a curved pipe with circular cross-section are reported. The Reynolds number based on the bulk velocity and the pipe diameter is equal to 5500, the radius of curvature of the pipe is equal to five pipe diameters. In the cross-section of the pipe a strong secondary motion is observed. This motion consist of counter rotating vortices at both sides of the symmetry plane, which are known as Dean cells. It is shown that in the turbulent case multiple solutions of governing equations exist. The output of an electromagnetic flow meter on this flow will be calculated, and compared with the results for a straight pipe.

4.1 Introduction

In this chapter we will use direct numerical simulation to study laminar and turbulent flow in pipe bends. This flow is of practical importance, because almost any flow appliance or equipment contains pipes with bends. Data on various parameters for such curved pipes are required for a number of applications. For instance, the increase in pressure loss due to a bend must be known in order to calculate the total pressure loss of a pipeline. Curved pipes are also used in heat exchangers where the secondary flow which occurs in the cross section of the bend, is used to transport heat from the pipe wall to the core of the pipe, so that the efficiency of the heat-exchanger is increased. It is obvious that a good knowledge of the strength of these secondary motions is crucial for the design of such a heat exchanger. Within the
context of topic of this thesis, namely electromagnetic flowmetering, curved pipes are also interesting because the secondary motion will also have an influence on the output of an electro magnetic flow meter located close to the bend, with as consequence that the indicated flowrate is incorrect. Flow in a curved pipe turns out to be also very interesting from a more theoretical point of view. The transition from laminar to turbulent flow in curved pipe flow appears not to be abrupt as in standard straight pipe flow. The laminar flow becomes first unsteady before the establishment of a fully developed turbulent flow. This transition from laminar flow to an unsteady state has been confirmed by the experimental work of Ramshankar & Sreenivasan (1988) and Webster & Humphrey (1997).

The general characteristics of curved pipe flow depends on the fact that the curvature leads to a centrifugal force. A (turbulent) flow field under influence of a centrifugal force undergoes a remarkable change in flow patterns as a result of the generation of secondary motions in the direction of the centrifugal force. This secondary motion can be characterised by the Dean number of the flow that we define here as:\(^1\)

\[ K = Re \sqrt{\frac{D}{R}}. \]  

(4.1)

Here \( Re \) is the Reynolds number based on the bulk velocity \( \bar{U} \), \( D \) the diameter of the pipe, and \( R \) is the radius of curvature. For loosely coiled pipes, i.e. \( R \) is large compared to \( D \), say \( R/D > 20 \), it appears that \( K \) is the sole parameter that determines the flow completely. In other words the Reynolds number can be replaced by the Dean number. However, for pipes with small \( R \) the flow must be characterised by both the Dean and Reynolds numbers.

Analytical solutions for laminar flow in curved pipes with low Dean numbers exists (see e.g., Van Dyke (1978)). In the literature also several numerical solutions are reported for laminar flows with higher Dean numbers. Most of these results are summarised in the review articles by Berger, Talbot & Yao (1983) and Ito (1987). In contrast, the number of publications on turbulent curved pipe flow is limited, and in the following we will shortly review this work.

Adler (1934) has performed some measurements and calculations on the fully-developed turbulent flow in loosely curved pipes at low Reynolds numbers. In this study only mean flow profiles were reported. Later, Ito (1959) did experiments to estimate the friction factor as a function of the curvature. In Rowe (1970) experimental results are presented for the flow in a curved pipe with a Reynolds number of \( 2.36 \times 10^5 \) and a Dean number of 68120. In this study it is shown that the axial velocity profile is completely altered and that the flow is fully de-

---

\(^1\)In the literature several definitions for the Dean number are used. Here we use the Dean number defined by Ito (1987).
veloped after an entrance length of approximately 18 diameters. Anwer, So & Lai (1989) and Anwer & So (1993) have carried out some measurements on the developing flow through a U-bend. The Dean numbers of these experiments are quite high and data on only a limited number of quantities are reported. Recently, Webster & Humphrey (1997) studied the transition to turbulence of a flow in a helical coil. In their paper some experimental data at low Reynolds numbers has been presented.

Turning to numerical modelling we can mention the articles of Patankar et al. (1974), Lai, So & Zhang (1991) and Boersma & Nieuwstadt (1995). Patankar et al. (1974) have used a $k - \epsilon$ closure model to resolve the parabolized Navier-Stokes equations. In Lai, So & Zhang (1991) also the parabolic Navier-Stokes equations are resolved using a Reynolds stress model instead of the $k - \epsilon$ model. In Boersma & Nieuwstadt (1995) the problem has been resolved for rather high Reynolds numbers using Large Eddy Simulation. In all these studies it is shown that strong secondary circulations are present in the cross section of the pipe. However, in none of these studies data is presented on turbulent statistics.

In this chapter we will consider the simulation of a fully developed turbulent flow in a curved pipe, with DNS, i.e. without any turbulence modelling. We have chosen for the fully developed curved pipe flow because in that case we can use periodic conditions for in and outflow section of the pipe. Such conditions are straightforward to implement, whereas the correct formulation of in- and outflow boundary conditions under general flow conditions is still far from clear. In the simulations the whole pipe cross section is computed without use of the symmetry which will be present in the plane of curvature. We will present data on various turbulence parameters, such as the turbulence intensities and the Reynolds stresses and discuss how they are influenced by the curvature. The results of the computation are used to estimate the effect of curvature on the reading of an idealised electromagnetic flowmeter. This gives the opportunity to estimate the error in reading of the flowmeter due to effects of the secondary motion in the curved pipe.

### 4.2 Governing equations

In this section we will present the governing equations for the flow in a curved pipe. The natural choice for a coordinate system for this geometry is a toroidal system. The details of such a system can be found in Morse & Feshbach (1953). In Figure 4.1 we have sketched such a coordinate system. The orthogonal metric of this system is given by Germano (1989):

$$
\mathbf{dx} \cdot \mathbf{dx} = dr^2 + (rd\phi)^2 + (1 + \kappa r \cos \phi)^2 ds^2,
$$

(4.2)
Figure 4.1: The coordinate system used, $u, v, w$ denote the velocity components in the radial, circumferential and axial direction, and $R$ is the radius of curvature. Plane I and II denote the planes in which we will present most of the results.

where $dr$, $d\phi$, and $ds$ are the infinitesimals in the radial, circumferential and axial direction, and $\kappa$ stands for the inverse radius of curvature, i.e. $\kappa = D/R$. With the metric given by equation (4.2) one can obtain the scale factors, as discussed in chapter two:

$$
\begin{align*}
    h_1 &= 1, \\
    h_2 &= r, \\
    h_3 &= 1 + \kappa r \cos \phi.
\end{align*}
\tag{4.3}
$$

With these scale factors we can derive the continuity and Navier-Stokes equations in conservative form, according to the equations given in chapter two. The continuity equation becomes:

$$
\frac{\partial}{\partial r} [rh_3 u] + \frac{\partial}{\partial \phi} [h_3 u] + \frac{\partial}{\partial s} [ru] = 0,
\tag{4.4}
$$

where $u$ the velocity in radial ($r$) direction, $v$ the velocity in tangential ($\theta$) direction, and $w$ the velocity in axial ($s$) direction. The Navier Stokes equations in conservative form become:

$$
\begin{align*}
    \rho & \left[ \frac{\partial u}{\partial t} + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 u^2] + \frac{\partial}{\partial \phi} [h_3 uv] + \frac{\partial}{\partial s} [ruu] \right) - \frac{v^2}{r} - \frac{w^2 \kappa \cos \phi}{h_3} \right] =  \\
    - \frac{\partial p}{\partial r} & + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 \tau_{rr}] + \frac{\partial}{\partial \phi} [h_3 \tau_{r\phi}] + \frac{\partial}{\partial s} [\tau_{rs}] \right) - \\
    \frac{\tau_{\phi\phi}}{r} & - \frac{\tau_{ss} \kappa \cos \phi}{h_3},
\end{align*}
\tag{4.5}
$$
\[ \rho \left[ \frac{\partial v}{\partial t} + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 vu] + \frac{\partial}{\partial \phi} [h_3 v^2] + \frac{\partial}{\partial s} [rvw] \right) + \frac{uv}{r} + \frac{w^2 \kappa \sin \phi}{h_3} \right] = \]

\[ - \frac{1}{r} \frac{\partial p}{\partial \phi} + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 \tau_r \rho] + \frac{\partial}{\partial \phi} [h_3 \tau_{r \phi}] + \frac{\partial}{\partial s} [r \tau_s] \right) + \frac{\tau_{r \phi}}{r} + \frac{\tau_{s s} \kappa \sin \phi}{h_3}, \]

(4.6)

\[ \rho \left[ \frac{\partial w}{\partial t} + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 wu] + \frac{\partial}{\partial \phi} [h_3 wv] + \frac{\partial}{\partial s} [rw^2] \right) + \frac{w \kappa}{h_3} [u \cos \phi - v \sin \phi] \right] = \]

\[ - \frac{1}{h_3} \frac{\partial p}{\partial s} + \frac{1}{rh_3} \left( \frac{\partial}{\partial r} [rh_3 \tau_r s] + \frac{\partial}{\partial \phi} [h_3 \tau_{r \phi} s] + \frac{\partial}{\partial s} [r \tau_{s s}] \right) + \frac{\tau_{r s} \kappa \cos \phi}{h_3} - \frac{\tau_{\phi s} \kappa \sin \phi}{h_3}. \]

(4.7)

where \( \rho \) is the mass density of the fluid, and \( \tau_{ij} \) the symmetric viscous stress tensor with components

\[ \tau_{rr} = \nu \left[ 2 \frac{\partial u}{\partial r} \right], \]

(4.8)

\[ \tau_{r \phi} = \nu \left[ \frac{1}{r} \frac{\partial u}{\partial \phi} + r \frac{\partial}{\partial r} \left( \frac{v}{r} \right) \right], \]

(4.9)

\[ \tau_{r s} = \nu \left[ \frac{1}{h_3} \frac{\partial u}{\partial s} + h_3 \frac{\partial}{\partial r} \left( \frac{w}{h_3} \right) \right], \]

(4.10)

\[ \tau_{\phi \phi} = \nu \left[ 2 \left( \frac{u}{r} + \frac{1}{r} \frac{\partial v}{\partial \phi} \right) \right], \]

(4.11)

\[ \tau_{\phi s} = \nu \left[ \frac{r}{h_3} \frac{\partial}{\partial s} \left( \frac{v}{r} \right) + \frac{h_3}{r} \frac{\partial}{\partial \phi} \left( \frac{w}{h_3} \right) \right], \]

(4.12)

\[ \tau_{s s} = \nu \left[ 2 \left( \frac{u \kappa \cos \phi}{h_3} - \frac{v \kappa \sin \phi}{h_3} + \frac{1}{h_3} \frac{\partial w}{\partial s} \right) \right]. \]

(4.13)

The equations are written in conservative form for numerical convenience, in view of the use of the finite volume method. In the sequel we will use a dimensionless form of the equations given above. The equations are nondimensionalised with the pipe diameter \( D \), the mean friction velocity \( U_* \), and time \( D/U_* \) as scale factors. The mean friction velocity \( U_* \) is defined here as:

\[ U_* = \frac{1}{2\pi} \left( \int_0^{2\pi} u_2^2(\phi) d\phi \right)^{1/2}, \]

(4.14)
where $u_*$ is the local friction velocity which is by definition equal to the square root of the local wall shear stress divided by the mass density of the fluid. For the dimensionless distance to the wall we will use $Y^+$ and $y^+ (Y^+ \text{ when we scale with } U_* \text{ and } y^+ \text{ when scaled with } u_*)$.

The flow is forced with a constant pressure gradient in the axial direction. This pressure gradient can be calculated from a simple force balance:

\[
\frac{1}{h_3} \frac{\partial P}{\partial s} = \frac{4\rho U_*^2}{D}.
\]

This forcing is almost the same as the one used in chapter three.

### 4.3 Results for Laminar flow

Fully developed laminar flow in curved pipes has been studied extensively in the past, by for instance, Collins & Dennis (1975) and Dennis & Ng (1982). In these articles a different dimensionless number is used. This number which is further on denoted by $D$ is in our notation equal to

\[
D = \frac{1}{2} Re_*^2 \sqrt{\frac{D}{2R}}.
\]  

(4.15)

In which $Re_*= U_*/D/\nu$. $D$ is related to the Dean number $K$ via the bulk velocity, which is not known apriori. Collins & Dennis (1975) and Dennis & Ng (1982) present several results in the range $96 \leq D \leq 5000$ using both finite difference and Fourier methods. Using the accurate Fourier method Dennis & Ng (1982) find multiple solutions of the Navier-Stokes. One of the solutions shows a double vortex pattern in the cross plane of the pipe and the other one with four vortices. In both cases the Dean number $K$ based on the bulk velocity is almost the same. With our finite volume method we are able to reproduces both solutions given by Dennis & Ng (1982) which is illustrated in Figure 4.2. In the top figure we show a contour plot of the axial velocity profile copied from the paper of Dennis & Ng (1982) for $D=5000$ (i.e. the two vortex solution). In the lower we show the axial velocity profile obtained with our code. The agreement between the numerical results of Dennis & Ng (1982) and ours is good, especially for the position and the value of the maximal velocity. The difference between the results is probably caused by the plotting procedure employed by Dennis & Ng (1982).

Collins and Dennis (1975) have also compared their calculations for $D = 5000$ ($K = 369.5$) with experimental data obtained by Adler (1934). The Dean number of Adler’s experiment is equal to $K = 1930 \cdot \sqrt{1/50} = 273$ (Fig 19 of Adler (1934)) and not equal to 372 as stated by Collins & Dennis (1975). The difference between
Contour levels of vector $\vec{u} \cdot 0$

Figure 4.2: Contour plots of the axial velocity. Top: The results of Dennis & Ng (1982) for $D = 5000$. Bottom: Our result for $D = 5000$. 
Figure 4.3: The axial velocity profiles in the symmetry plane. Left: Computation of Collins and Dennis (1975) at \( K = 369.5 \) (line), and experiment of Adler(1934) at \( K = 273 \) (symbols). Right: Our computation at \( K = 273 \) (line) and the experimental data of Adler (1934) (symbols). In both cases the velocity is normalized with the bulk velocity.

the computation of Collins & Dennis (1975), and experiment is therefore quite large as can be seen from Figure 4.3 (left), which we have copied from the original paper of Collins & Dennis (1975). In Figure 4.3 (right) we compare results for the correct value \( K = 273 \) with the experimental data of Adler (1934). This figure shows excellent agreement between simulation and experiment.

In this section we have shown that our code reproduces the results of Dennis & Ng (1982) accurately. Furthermore we have shown that our simulation show excellent agreement with the experimental data of Adler (1934). Both these points give us confidence in our computer code and based on this observation we will extend our computations to turbulent flow, for which much less (experimental) data for comparison are available.

4.4 Results for Turbulent flow

In this section we will investigate turbulent flow in curved pipes. We will do this with the same computer code that we have used in the previous section for the laminar flow. We aim to perform a DNS, i.e. we have to resolve all scales of turbulence (including the Kolmogorov scales) on the computational grid, where the Kolmogorov length scale is given by equation (2.23). For curved pipe flow we expect that this length scale is the same or perhaps somewhat larger than in standard pipe flow due to the large secondary flow vortices. As a consequence the Kolmogorov
length scale will be the same or somewhat larger for curved pipe flow than for standard pipe flow \(^2\).

For a fully resolved DNS we need at least two grid points to resolve the Kolmogorov scale (Nyquist criterium). With the amount of grid points that we can use this is only possible for a Kolmogorov length scale, \(\eta\), which is rather large. As a consequence we can only calculate a turbulent flow with a rather low Reynolds number. For straight pipe flow, it is well known that the critical Reynolds number is approximately 2300. Above this number the flow can become turbulent. Experiments carried out by Taylor (1929) in a curved pipe show that the critical Reynolds number in this case can be a factor two or more larger, due to the stabilising effect of the centrifugal force on the flow. Therefore we aim to calculate a flow with a bulk Reynolds number larger than 5000.

4.4.1 Computational details

All the DNS runs are started with a velocity profile valid for a fully developed straight pipe:

\[
\begin{align*}
  w &= 2.5U_\ast \ln \left( \frac{yU_\ast}{\nu} \right) + 5.5 \quad \text{if} \quad yU_\ast/\nu \geq 11.6 \\
  w &= yU_\ast^2/\nu \quad \text{if} \quad yU_\ast/\nu < 11.6.
\end{align*}
\]

where \(y = D/2 - r\) is the coordinate measured from the wall. On this profile small random fluctuations are superimposed. After approximately four time scales \(D/U_\ast\) the flow becomes fully developed, i.e. turbulent statistics become independent of time and initial conditions. We note that for standard pipe flow this takes approximately \(8D/U_\ast\). The difference in time to reach stationarity can be attributed to the additional mixing by the secondary motion, which removes the transients from the initial conditions rapidly.

In Table 4.1 we have summarised some important quantities of the DNS of curved pipe flow (denoted by A, B and C) and of a DNS of standard pipe flow (simulation D) which has been calculated by Eggels \textit{et. al.} (1994). The computational grid in simulations A, B and C is non-uniform in the direction towards the wall, with the grid points slightly concentrated near the wall to resolve the near wall layer. For the standard pipe flow a uniform grid has been used and therefore, much more grid points are required to get the same resolution in the near wall region.

The statistics for the standard pipe flow simulation (run D) were computed over forty independent data fields by spatial averaging in the homogeneous tangential and axial direction. For the curved pipe flow only spatial averages in the axial

\(^2\text{A priory it is not possible to calculate the Kolmogorov length scale.}\)
Table 4.1: Some important quantities used in the DNS of turbulent flow. Here \(N\) denotes the number of grid points in the radial, tangential and axial direction respectively, \(Re_*\), the Reynolds number based on the mean friction velocity and pipe diameter, \(R\) the radius of curvature, \(K\) and \(D\) are Dean numbers defined by (1.1), (4.1), and \(L/D\) the length of the computational domain normalised with the pipe diameter.

<table>
<thead>
<tr>
<th></th>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
</tr>
</thead>
<tbody>
<tr>
<td>(N)</td>
<td>90 (\times) 160 (\times) 350</td>
<td>90 (\times) 160 (\times) 350</td>
<td>62 (\times) 128 (\times) 256</td>
<td>96 (\times) 128 (\times) 256</td>
</tr>
<tr>
<td>(Re_*)</td>
<td>460</td>
<td>460</td>
<td>460</td>
<td>360</td>
</tr>
<tr>
<td>(R)</td>
<td>5</td>
<td>5</td>
<td>5</td>
<td>(\infty)</td>
</tr>
<tr>
<td>(Re_b)</td>
<td>5470</td>
<td>5520</td>
<td>5500</td>
<td>5300</td>
</tr>
<tr>
<td>(K)</td>
<td>2445</td>
<td>2470</td>
<td>2460</td>
<td>0</td>
</tr>
<tr>
<td>(D)</td>
<td>33500</td>
<td>33500</td>
<td>33500</td>
<td>0</td>
</tr>
<tr>
<td>(L/D)</td>
<td>7.5</td>
<td>7.5</td>
<td>6.5</td>
<td>5</td>
</tr>
</tbody>
</table>

Direction can be used and therefore much more data fields are needed to obtain stable statistics. For the curved pipe flow we have used 80 data fields (80 data fields \(\approx\) 14Gb disk space!), which probably is still too few to get fully reliable statistics.

The simulations A and B, listed in Table 4.1, are performed for exactly the same \(Re_*\) and \(R\) however with different initial conditions. Simulation B was started from a pseudo random number generator\(^3\) and became stationary after approximately four time scales \(U_*/D\). The variances obtained from simulation B were not in agreement with earlier results (e.g., Boersma & Nieuwstadt (1995)). The computation was therefore repeated with as initial conditions a velocity field from run B with extra energy in the form of random noise added to it\(^4\). This resulted in a second stationary solution, i.e. simulation A, which shows the same behaviour as reported by Boersma & Nieuwstadt (1995) and Adler (1934). In the sequel of this chapter we will present some results for both stationary solutions and we will compare these results with data of the DNS for standard pipe flow and with the available experimental data of Webster & Humphrey (1997).

First, we will check if the length of the computational domain used in our simulations, is sufficient long to resolve all large scale turbulent motions. We will do

---

\(^3\)We have used a random number generator which generates uniform random numbers between \(w - a\) and \(w + a\). Where \(w\) is obtained from equation (4.16) and \(a = 0.05w\).

\(^4\)In this case the random number generator, generated random numbers between \(w - 0.075w\) and \(w + 0.075w\), where \(w\) is the average velocity obtained from the simulations.
this by calculating the longitudinal velocity correlation, defined by

$$\rho_{xx}(l) = \frac{\chi(r, \phi, s)\chi(r, \phi, s + l)}{\chi(r, \phi, s)^2}. \quad (4.17)$$

in which $\chi$ denotes a flow-variable. This correlation should be small halfway the computational domain, where small means $< 10\%$. This is interpreted as the influence of the periodic boundary conditions on the structures of turbulence, i.e. if this correlation is small then the influence of the periodic boundary conditions on the simulated flow will be small. In Figure 4.4 we have plotted these velocity correlations obtained from simulation A, for all three components, at the symmetry plane (plane I in Figure 4.1) both near the in- and outside of the bend and at a distance of $20Y^+$ from the wall. The correlations at halfway ($s = 3.75D$) of the computational domain are quite small near the outside of the bend and of the same order as for the standard pipe flow (not shown here). At the inside of the bend the correlations oscillate somewhat which is probably caused by too few statistically independent samples. Such oscillations are not present at the outside of the pipe bend. This can be explained as follows: a characteristic time scale for a turbulent flow in the near-wall region is: $D/u_*$ where $u_*$ is the local friction velocity. At the outside of the pipe $u_* > U_*$ and at the inside $u_* < U_*$. Thus, the local time scale of the turbulence at the inside is larger than at the outside. Later on we will see that the friction velocity at the outside of the pipe is almost five times larger than the friction velocity at the inside. The consequence of this is that the time scale at the inside is five times larger than the time scale at the outside, i.e. we need approximately five times more samples at the inside of the bend than at the outside to obtain the same accuracy in the statistics.

From the velocity correlations shown in Figure 4.4 and the fact that they reduce to a small value halfway the computational domain we conclude that the computational domain is long enough, and also that our results at the inside are possible not be complete converged in a statistical sense.

### 4.4.2 Mean flow quantities

In Table 4.1 we have listed the computed results for the bulk Reynolds and Dean number $K$ defined according to (4.1) (4.15). The differences between these numbers for simulations A,B and C is quite small, which suggests that the results may be considered as grid independent. In Figure 4.5 we show the mean velocity profiles obtained from the simulation A and B (curved pipe flow) in combination with simulation D (standard pipe flow) and the experimental results of Webster & Humphrey (1997). The Reynolds and Dean numbers of the experiment
Figure 4.4: The longitudinal velocity correlations at the symmetry plane. The figure shows the correlation which are calculated near the inside of the bend, $Y^+ = 20$, $y^+ = 6.4$, (top) and the second figure (bottom) shows the correlation near the outside of the bend ($Y^+ = 20$, $y^+ = 28.1$)
are 5400 and 1280 respectively. The Reynolds number of experiment and simulation is thus the same, but the Dean number is different.

The mean velocity profiles are presented in the planes I and II as defined in Figure 4.1. The difference in the axial velocity profiles for curved pipe flow and standard pipe flow is striking. From the figure it is clear that due to the centrifugal force the rapidly flowing parts of the flow, move to the outside of the bend while the slowly flowing fluid elements are forced to the inside of the bend. Consequently, the velocity is low at the inside and high at the outside of the bend. Furthermore, there is a significant difference between the axial velocity profiles from simulations A and B near the outside of the bend, to which we will come back later. The velocity profiles in plane II are symmetric and have a maximum close to the pipe wall. This velocity maximum is caused by the secondary motion which advects the high velocity present at the outside of the bend along the wall. The agreement between the experiment of Webster & Humphrey (1997) and simulation A is reasonable. The differences apparent in Figure 4.5 are probably caused by the difference in Dean numbers, i.e. the difference in magnitude of the centrifugal force between experiment and simulation. Furthermore, the experimental data indicate that simulation A is realistic and not just some kind of a numerical artifact.

In Figure 4.6 we show contour plots of the axial velocity profiles which follow from simulation A and B. This figure shows again clearly that the centrifugal force, displaces the rapid flowing parts of the fluid to the outside of the pipe and that there is only a difference between simulation A and B at the outside of the bend.
Figure 4.6: A contour plot of the axial velocity profile normalised with $U_\ast$. The top figure is for case A and the bottom figure for case B (in both cases the outside of the curvature is to the right).
Figure 4.7: The local friction velocity $u_*$ as a function of the circumferential angle $\phi$ normalised with the friction velocity averaged over the perimeter of the pipe, for the curved pipe flow calculations A, B, and for standard pipe flow.

Next, we consider the local friction velocity $u_*(\phi)$ (at the pipe wall) which is by definition equal to

$$u_*(\phi) = \sqrt{\tau_{rs}(D/2, \phi)} \quad \text{with} \quad \tau_{rs}(D/2, \phi) = \nu \frac{\partial w(\phi)}{\partial r}.$$  \hspace{1cm} (4.18)

In Figure 4.7 we have plotted $u_*$ as a function of the circumferential angle $\phi$ ($\phi = 0$ corresponds to the outside of the bend). The figure shows that the wall shear stress is high at the outside of the bend and low at the inside. Above we have already interpreted the result in terms of the time scale of turbulence, which is small near the outside and large near the inside of the bend. It follows from Figure 4.7 that the time scale of the turbulence $D/u_*$ near the inside of the bend is almost five times larger than near the outside. Furthermore it can be seen that the time scale in plane II, i.e. at $\phi = \pi/2$ is the same as for straight pipe flow, for which the time scale is $D/U_*$. From Figure 4.7 we can also deduce that the distance, in wall units, along the inside of the bend is only $460 \cdot 7.5(D/2R) \cdot 0.35 = 1090y^+$. This is quite short compared to the length of an ordinary low speed streak (typical 1000 wall units) and this partly explains the rather higher auto-correlation near the inside of the bend (e.g. section 4.4.1).
4.4.3 Secondary Motion

We have already mentioned that there is a strong secondary motion present in the cross plane of the pipe. In this section we will study this motion in more detail. The secondary motion can be characterised by a stream function $\psi$ which is defined according to:

$$
\psi(r, \phi) = -\int_r^{D/2} v(r, \phi) \, dr
$$

(4.19)

Here, we haven taken the stream function zero at the wall. In Figure 4.8, the stream function has been plotted, for both simulations A and B. The distance between the streamlines is a measure for the secondary velocity. The figure shows that the secondary velocity is very strong near the pipe wall close to plane II. Furthermore, it follows that the secondary velocities are much larger at the inside of the bend than at the outside.

The strong vortices which appear in Figure 4.8 near the inside of the bend, are regular Dean vortices which are also present in laminar curved pipe flow (e.g., Collins & Dennis (1975)). Comparison of the contour plots of the secondary motion obtained for simulations A and B shows a small difference near the outside of the bend, which corresponds with the difference which we have already noted in Figure 4.5. This difference can explained qualitatively as follows. The pressure gradient in the radial direction in the symmetry plane can be approximated as:

$$
\frac{\partial P}{\partial r} \approx \frac{\rho w^2}{R \cdot r}.
$$

(4.20)

which is a balance between centrifugal acceleration and pressure gradient. From Figure 4.5 one can see that the toroidal velocity for case B near the outside of the bend is approximately 10% larger than for case A. Consequently the pressure gradient $\partial P/\partial r$ will be larger for case B. This implies that the outward velocity in the mid plane for case B is smaller than for case A, and this explains the small difference in secondary flow patterns near the mid plane. The velocity in the vertical (take along plane II) direction near the pipe centreline in Figure 4.8 is caused by the Coriolis force, which results from the rotation induced by the Dean cells.

4.4.4 Turbulent statistics

In this section we will investigate how the curvature, i.e. the centrifugal force, influences the turbulent statistics. To compute the statistics we introduce the standard Reynolds decomposition

$$
u = \bar{u} + u', \quad v = \bar{v} + v', \quad w = \bar{w} + w', \quad \text{and} \quad p = \bar{p} + p',
$$

(4.21)
Figure 4.8: The secondary motion in the cross plane of the pipe defined by equation (4.19). The top Figure shows the results obtained from simulation A and the bottom Figure the results obtained from simulation B. In both case the outside of the curvature is to the right.)
where a bar denotes an average in the homogeneous axial direction and in time. A prime denotes a deviation of this average. When we apply the Reynolds decomposition given above to equation (4.7) this leads to the following equation for the mean flow quantities in the axial direction:

\[
\frac{\partial}{\partial r} [r h_3 (\bar{u} \bar{w} + \bar{u}' \bar{w}')] + \frac{\partial}{\partial \phi} [h_3 (\bar{u} \bar{w} + \bar{u}' \bar{w}')] + r \kappa \bar{w} (\bar{u} \cos \phi - \bar{v} \sin \phi) = 4 r U_*^2 / D + \frac{\partial r h_3 \bar{\tau}_{rs}}{\partial r} + \frac{\partial h_3 \bar{\tau}_{\phi\phi}}{\partial \phi} + r \kappa (\bar{\tau}_{rs} \cos \phi - \bar{\tau}_{\phi\phi} \sin \phi).
\]

(4.22)

In the symmetry plane ($\phi = 0$) some terms vanish, viz.

\[
\bar{v} = \frac{\partial \bar{u}}{\partial \phi} = \frac{\partial \bar{w}}{\partial \phi} = 0,
\]

(4.23)

and equation (4.22) reduces to

\[
r h_3 \bar{u} \frac{\partial \bar{w}}{\partial r} + \frac{\partial}{\partial r} [r h_3 \bar{u}' \bar{w}'] + \frac{\partial}{\partial \phi} [h_3 \bar{u}' \bar{w}'] + r \kappa \bar{u} \bar{w}' \cos \phi - r \kappa \bar{u}' \bar{w} \sin \phi = 4 r U_*^2 / D + \frac{\partial}{\partial r} [r h_3 \bar{\tau}_{rs}] + r \kappa \bar{\tau}_{rs},
\]

(4.24)

where we have used to continuity equation to rewrite some of the partial derivatives in equation (4.22). In the core region of the pipe we can neglect the viscous terms in equation (4.24) and this leads to the following equation for the mean flow in plane I

\[
\frac{h_3}{r} \bar{u} \frac{\partial \bar{w}}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} [r h_3 \bar{u}' \bar{w}'] + \frac{1}{r} \frac{\partial}{\partial \phi} [h_3 \bar{u}' \bar{w}'] + \kappa \bar{u} \bar{w} + \kappa \bar{u}' \bar{w}' = 4 U_*^2 / D.
\]

(4.25)

In Figure 4.9a and b we have plotted the five terms on the left hand side of the equation (4.25), normalised with $U_*^2 / D$ The sum of the five terms in Figure 4.9a and 4.9b is almost constant in the core region and equal to $4 U_*^2 / D$. This value is equal to the forcing term which we have described previously. In both case $\bar{u} \partial \bar{w}/\partial r$ is by far the largest term. This term mainly balances the imposed pressure gradient in the axial direction. This balance is in rather contrast to standard pipe or channel flow where the pressure gradient is balanced by the Reynolds stress $\tau^{-1} \partial (r \bar{u}' \bar{w}') / \partial r$. In principle the conclusion based on Figure 4.9 is only valid in the symmetry plane however similar behaviour will be observed in the other parts of the flow.

The balance between $\bar{u} \partial \bar{w}/\partial r$ and the imposed pressure gradient partly explains the large difference between parallel turbulent flow and mildly curved turbulent flow (e.g., Bradshaw (1987)), i.e. a small curvature ratio will give also a small radial velocity, but $\bar{u} \partial \bar{w}/\partial r$ can be large, so that

\[
\bar{u} \partial \bar{w}/\partial r \approx \frac{\partial P}{\partial s}, \text{ and } \bar{u}' \bar{w}' \ll \frac{\partial P}{\partial s}
\]
Turbulent flow in curved pipes

Figure 4.9: The terms at the right hand side of equation (4.25). top: simulation A, bottom: simulation B. I: \( rh_3 \overline{uu'}, \) II: \( r^{-1} \frac{\partial}{\partial r} [rh_3 \overline{uw'}], \) III: \( r^{-1} \frac{\partial}{\partial s} [h_3 \overline{uw'}], \) IV: \( \kappa \overline{u}, \) V: \( \kappa \overline{u}. \) (The right part of the figure is a blow up of the left figure.)

Furthermore, Figure 4.9b shows that the derivatives of the Reynolds stresses \( \overline{uu'} \) and \( \overline{uw'} \) for case B remain small near the outside of the bend, in contrast to the values of these quantities for case A, shown in Figure 4.9a. Thus in case A the momentum \( \frac{\partial P}{\partial s} \) is balanced by the Reynolds stresses and \( \overline{u} \frac{\partial \overline{w}}{\partial r} \) while in case B the momentum is balanced by \( \overline{u} \frac{\partial \overline{w}}{\partial r} \) only. The absence of the Reynolds stress in calculation B, could imply that the flow is laminar, but as will be shown in the sequel this is not the case.

Next we will compare curved pipe flow with straight pipe flow. For a fully developed straight turbulent pipe flow the following simple force balance can be derived

\[
\frac{1}{r} \frac{\partial}{\partial r} \left( r \nu \frac{\partial w}{\partial r} + r \overline{u'w'} \right) = \frac{1}{\rho} \frac{\partial P}{\partial s} = 4u_*^2 / D.
\]

Near the core of the pipe the viscous term is negligible and \( \overline{u'w'} \) is a linear function of \( r \). For curved pipe flow we find near the centreline of the pipe the following dif-
ferential equation for $\overline{u'w'}$, under the assumption that viscous effects are negligible

$$h_3\overline{u'} \frac{\partial \overline{w'}}{\partial r} + \frac{1}{r} \frac{\partial}{\partial r} \left[r h_3 \overline{u'w'}\right] + \kappa \overline{u'w'} = 4U_*^2/D + \text{other terms}$$

When $\kappa = 0$ ($h_3 = 1, \overline{u'} = 0$) we obtain the same equation as for straight pipe flow. For a non-zero curvature the differential equation changes type and consequently also the form of the solution. As mentioned before the Reynolds stress $\overline{u'w'}$ in a straight pipe will be linear in the core region, for curved pipe flow the Reynolds stress $\overline{u'w'}$ will be proportional to $\exp(-\kappa r)$, the Reynolds stress will thus decrease when $r$ increases (e.g. Figure 4.12 $\phi = 0^\circ$).

In Figure 4.10 and 4.11 we show for both the straight and curved pipe the root-mean-square (rms) values of the fluctuating velocity components normalised with the mean friction velocity $U_*$. The rms values for the curved pipe at the inside of the bend in the symmetry plane are for simulations A and B almost the same. In both cases the radial and tangential velocity fluctuations are larger than the axial velocity fluctuations. Near the outside of the bend there is however a large difference in these rms profiles. Simulation A shows a large axial velocity fluctuation and quite small radial and tangential fluctuations which is in agreement with earlier observations at higher Reynolds numbers by us (e.g., Boersma & Nieuwstadt (1995)) and by Anwer, So & Lai (1989). In contrast simulation B shows very small velocity fluctuations near the outside of the pipe, especially for the radial velocity component. The symbols in Figure 4.10 denote the experimental data obtained by Webster & Humphrey (1997). Quantitatively, the agreement between these experimental data and DNS is not very good, which is probably due to the difference in the Dean number. However, qualitatively both the experiments and DNS are quite similar, at least much more than in comparison with case A. Both results show a peak in axial rms near the pipe centreline, which corresponds to the inflexion point in the axial velocity, e.g. Figure 4.5.

In plane II, we also observe a difference between the root-mean-square profiles obtained from simulation A and B. However, this difference is much less pronounced than near the outside of the bend. The root-mean-square profiles near the pipe wall have a similar shape as in a standard pipe or channel flow. However there is a large difference in the numerical values.

In Figure 4.12 and 4.13 we show the profiles of the $\overline{u'w'}$ Reynolds stress at various angles $\phi$ (see Figure 4.1). The Reynolds stress obtained from run A is small in the core region of the pipe and quite large near the outside of the pipe, for all circumferential positions. The Reynolds stress for calculation B is much smaller than for case A, which is consistent with the results for the rms of the velocity fluctuations discussed above.

It is clear from the Figures 4.10, 4.11, 4.12, 4.13 that the turbulent kinetic energy
Figure 4.10: The root-mean-square values of the velocity fluctuations in curved pipe flow, in plane I. The top Figure is obtained from solution A and the bottom figure obtained from solution B.
Figure 4.11: The root-mean-square values of the velocity fluctuations in curved pipe flow (plane II). The top Figure is for solution A and the bottom Figure for solution B.
Figure 4.12: The $\overline{u'w'}$ Reynolds stress as a function of the radial position, for various angles of $\phi$. (Calculation A)

Figure 4.13: The $\overline{u'w'}$ Reynolds stress as a function of the radial position, for various angles of $\phi$. (Calculation B)
for case B is much smaller than for case A. The consequence is a less strong
turbulent transport which as a consequence leads to a somewhat higher bulk velocity
for case A, e.g. Table 4.1

4.4.5 Dissipation

In this section we will consider the dissipation of kinetic energy, which is defined as:

$$\varepsilon = -\frac{\partial u_i^j}{\partial x_j}$$

(4.26)

The dissipation can be used to calculate the local Kolmogorov length scale in the flow, i.e. the smallest length scale in the flow. This gives us the opportunity to
investigate if our DNS is fully resolved. In Figure 4.14, 4.15 we show contour plots
of the dissipation of kinetic energy for calculations A and B. We find again that
there is a large difference between the results for both calculations. The maximum
dissipation for calculation A is equal to $131 U_\ast^3 / D$. The Kolmogorov length scale
is given by equation (2.23). Using this equation we obtain for calculation A the
following Kolmogorov length scale:

$$\eta = (\nu^3 / \varepsilon)^{1/4} = 0.0030D$$
Figure 4.15: The dissipation of kinetic energy obtained from calculation B

which in dimensionless wall units becomes equal to $\eta_* = 1.4$. Based on this value we find that the maximum grid spacing, according to the Nyquist, should be

$$\Delta_{max} \leq \frac{1.4}{2} = 0.7 y^+,$$

close to the wall. This is the case for the radial direction, where $\Delta r \approx 0.6 y^+$, but not for the circumferential and axial direction where $r\Delta \phi \approx 9 y^+$ and $\Delta z = 9.8 y^+$. However, this resolution, in dimensionless units, is comparable to the resolution used by several other researchers (see for instance Eggels et. al. (1994), Kim, Moin & Moser (1987), Gavrilakis (1992)). Furthermore, we have seen that the results obtained from run C and run A show only minor differences, so we do not expect different results when we increase the number of grid points in the periodic directions.

For simulation B the maximum dissipation is approximately $28U^3/D$, and the Kolmogorov length scale is equal to $2.2 y^+$. So this simulation will be somewhat better resolved than simulation A.

### 4.4.6 Production

In this section we will give some results for the production of turbulent kinetic energy. The production of kinetic energy is defined as:

$$P = (P_{11} + P_{22} + P_{33})$$
74 Chapter 4

Figure 4.16: An isosurface plot of the production of kinetic energy obtained from simulation A.

In which

\[ P_{ij} = -\bar{u}_i u_j \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right). \]

In Figure 4.16 and 4.17 we show an isosurface plot of the production of kinetic energy obtained from simulation A and B respectively. Again there is a large difference between the two solutions. Calculation A shows a rather high production of kinetic energy at the outside of the curvature. For case B the maximum is located at the pipe wall in plane II. From the Figures 4.14, 4.16 and 4.15, 4.17 it follows that the frequently used assumption that production equals dissipation holds reasonable well for this flow.

4.4.7 Summary of the flow simulation

In previous sections we have reported a number of quantities obtained from several Direct Numerical Simulations in a curved pipe. One of these simulations (A) shows similar behaviour as we have found previously with help of LES, i.e., the turbulence is suppressed at the inside of the bend and enhanced at the outside of the bend. A second simulation (B) at exactly the same Reynolds number and curvature parameter shows a totally different behaviour. In this simulation the flow near the outside of the bend is almost laminar while at the inside of the bend the flow is still
turbulent. The existence of such multiple solutions for a turbulent curved pipe flow has not been reported in the literature, at least to our knowledge. The multiple solutions which occur in the statistics must be interpreted as a different turbulent structure. It should be mentioned that in a related geometry, i.e. the turbulent flow in a curved duct multiple solutions have been observed experimentally, Ito (1987). Therefore it is not surprising that such multiple solutions are also present in curved pipe flow.

4.5 Response of an electromagnetic flowmeter on curved pipe flow

In the last section of this chapter we will use the DNS results presented before to calculate the response of an electromagnetic flowmeter on curved pipe flow and compare this with the response in a straight pipe. The background is the fact that an electromagnetic flowmeter is only calibrated for a straight pipe. However, in practical situations one is frequently confronted with the fact that the flowmeter has to be installed near a pipe bend (in general one assumes that the effect of the bend has vanished after five diameters). To get an estimation of the effect we consider here the case of an idealised electromagnetic flowmeter in a fully developed
curved pipe flow.

For the right hand side of the flowmeter equation (2.15) we have to calculate the vorticity. The vorticity in toroidal coordinates is given by (e.g. chapter 2):

\[
\omega_r = \frac{1}{r h_3} \left( \frac{\partial [h_3 w]}{\partial \phi} - \frac{\partial [rv]}{\partial s} \right),
\]

(4.27)

\[
\omega_\phi = \frac{1}{h_3} \left( \frac{\partial u}{\partial s} - \frac{\partial [h_3 w]}{\partial r} \right),
\]

(4.28)

\[
\omega_s = \frac{1}{r} \left( \frac{\partial [rv]}{\partial r} - \frac{\partial u}{\partial \phi} \right).
\]

(4.29)

For the magnetic field \( \mathbf{B} \) we will use again a homogeneous magnetic field:

\[
\mathbf{B} = (B \cos \phi, B \sin \phi, 0)
\]

and we will also consider a magnetic field with an orientation perpendicular to the one given above:

\[
\mathbf{B} = (B \cos(\pi/2 + \phi), B \sin(\pi/2 + \phi), 0)
\]

In Figure 4.18 and 4.19 we show contour plots of the calculated electric potential, obtained with the velocity profiles from DNS A. Comparing this result with the electrical field computations for a straight pipe flow given in chapter three we find that the difference between the induced electric potential in curved and straight pipe flow is much smaller than the difference in the velocity fields, which we have considered in the previous section. Furthermore, we observe that the gradient in the electric potential, i.e. the distance between the contour lines, is proportional to the axial velocity for the case with \( \mathbf{B} = (B \cos \phi, B \sin \phi, 0) \) given in Figure 4.18, while this is not the case for the other magnetic field.

The maximum potential differences are 11.52\( BDU_* \) and 12.449\( BDU_* \), respectively. The bulk velocity obtained by direct integration of the velocity field is equal to 11.9\( U_* \) (calculation A). In chapter three we have seen that for the case of fully developed straight pipe flow the induced electric potential difference is within 0.5% the same as calculated from the bulk velocity. So when an electromagnetic flowmeter is placed in a bend or directly after a bend the error in the reading of the flowmeter is

\[
\text{err} = \frac{|11.52 - 11.9|}{11.9} \times 100\% = 3.2\% \quad \text{or}
\]

\[
\text{err} = \frac{|12.45 - 11.9|}{11.9} \times 100\% = 4.6\%.
\]

From this we can draw the conclusion that for a flowmetering device placed in or directly after a bend with a magnetic field perpendicular to the plane of curvature
Figure 4.18: The induced electric potential in a curved pipe, with $\mathbf{B} = (B \cos \phi, B \sin \phi, 0)$.

Figure 4.19: The induced electric potential in a curved pipe, with $\mathbf{B} = (B \cos(\pi/2 + \phi), B \sin(\pi/2 + \phi), 0)$. 
gives a slightly better reading than a device with a magnetic field parallel to the plane of curvature.

With help of the DNS data we can also calculate an "optimal angle" of the magnetic field, i.e. the direction of the magnetic field for which the difference between the reading, of a flowmeter in a curved pipe and a straight pipe is minimal.

Let us assume a magnetic field given by

\[ \mathbf{B} = (\cos(\alpha + \phi), \sin(\alpha + \phi), 0) \]

where \( \alpha \) is the angle between the magnetic field and plane I. In Figure 4.20 we show the maximum difference in the electric potential for a curved pipe flow, by the magnetic field given above with \( \alpha \) varying between 0..2\( \pi \). The dashed line in this figure denotes the bulk velocity, 11.9 \( U_* \), which has been obtained by direct integration of the axial velocity field. In Figure 4.21 we show the induced potential for calculation \( B \). The dashed line denotes again the bulk velocity obtained from direct integration of the velocity field. In both cases it is clear that a magnetic field with an orientation parallel or perpendicular to direction of curvature gives a relative large error in the reading of the flowmeter. The error is minimal for a magnetic field which makes an angle of approximately \( \pm57 \) degrees (1 rad) with the plane of curvature.

In practise electromagnetic flowmeters do not have a homogeneous magnetic field. However we expect that for a more realistic magnetic field, this result is still valid.

4.6 Conclusion

In this chapter we have used direct numerical simulation to study fully developed turbulent flow in a curved pipe, with a small radius of curvature. We have reported some results for laminar flow which shows perfect agreement with the existing experimental and numerical studies. The Reynolds number and Dean number for the turbulent case are 5500 and 2450, respectively. It is shown that at this low Reynolds number, there are two stationary solutions of the Navier-Stokes equations. One of these solutions is a fully turbulent one, i.e., the flow is everywhere turbulent in the cross-section of the pipe (except the viscous sub layer). The results of another simulation (B) shown an almost laminar flow near the outside of the bend and turbulent near the inside of the bend. We expect that this second solution can only be found for a small range of Reynolds and Dean numbers at which transition can occur.

The existence of a solution which is partly laminar an partly turbulent is maybe and explanation for the gradual transition from laminar to turbulent flow in a curved
Figure 4.20: The maximum induced electric potential difference as function of the angle between the homogeneous magnetic field and the plane of curvature (Calculation A).

Figure 4.21: The maximum induced electric potential difference as function of the angle between the homogeneous magnetic field and the plane of curvature (Calculation B).
pipe which is found in experiments Ramshankar & Sreenivasan (1988), Taylor (1929), Webster & Humphrey (1997).

Furthermore, we have shown that the reading of a flowmeter placed in or directly after the bend gives an incorrect reading of about 4% with respect to the reading in a straight pipe. For an electromagnetic flowmetering device this is a relative large error. It is also shown that the most reliable reading is obtained when the angle between the homogeneous magnetic field and the plane of curvature is approximately fifty seven degrees.
Chapter 5

Analogy between resistive MHD and viscous fluids

From the literature, (see for instance Shercliff (1965), Moreau (1990)), it is known that there is a close analogy between the flow of a viscous fluid and the flow of resistive plasma. In this chapter we will first give the theoretical background for this analogy. Subsequently we will use this analogy to investigate the behaviour of a resistive plasma in a tokamak. We will do this by studying the related hydrodynamical problem.

5.1 Introduction

It is a well know fact that there exists a remarkable analogy between the equations governing resistive magneto hydrodynamics and viscous fluids, e.g. Shercliff (1965) Moreau (1990). The equation which governs the magnetic field in a resistive plasma has been derived in chapter two, e.g. equation (2.11):

\[
\frac{\partial \mathbf{B}}{\partial t} = \nabla \times (\mathbf{u} \times \mathbf{B}) + \frac{1}{Re_m} \nabla^2 \mathbf{B}.
\]

The Navier-Stokes equations for a fluid flow have also been introduced in chapter two. When we take the curl of the Navier-Stokes equations we obtain the following equation for the vorticity in the fluid:

\[
\frac{\partial \omega}{\partial t} = \nabla \times (\mathbf{u} \times \omega) + \frac{1}{Re} \nabla^2 \omega. \tag{5.1}
\]
In which \( \omega \) is the vorticity vector which is by definition equal to \((\nabla \times \mathbf{u})\). The analogy between \( \mathbf{B} \) and \( \omega \) will be clear from the equations given above. For a resistive plasma the magnetic Reynolds number will be of \( \mathcal{O}(10^4..10^6) \) which is comparable to the Reynolds number which can be found in pipe flow. So there exists an analogy between the vorticity, \( (\omega) \), in a normal fluid and the magnetic induction, \( \mathbf{B} \), in resistive magneto hydrodynamics. However, this analogy is not perfect, because \( \omega \) is related to the velocity \( \mathbf{u} \), \( (\omega = \nabla \times \mathbf{u}) \) in a way that \( \mathbf{B} \) is not. Milligen (1991) included a non-conservative body force in equation (5.1). So equation (5.1) becomes
\[
\frac{\partial \omega}{\partial t} = \nabla \times (\mathbf{u} \times \omega + \mathbf{F}) + \frac{1}{Re} \nabla^2 \omega
\]
(5.2)
where \( \mathbf{F} \) is the non-conservative body force. The equations are still not identical but are now of comparable complexity.

So we now have the interesting situation that processes occurring in an ordinary fluid have relevance to resistive MHD. Hereby, it should be noted that the hydrodynamic problem is less complicated than the MHD problem. Because in the hydrodynamic problem there are four unknowns (the three components of the velocity and the pressure), while in the MHD case the three components of the magnetic induction \( \mathbf{B} \) are additional unknowns.

The analogy outlined above will be used as starting point to study a problem occurring in plasma physics. This problem is the geometric resonances which occurs in tokamak plasma's, which leads eventually to plasma turbulence.

5.2 The geometry

In this section we will describe the geometry in which our calculations have been carried out. This geometry is related to the geometry used in a tokamak plasma fusion reactor. Plasma experiments are frequently carried out in a tokamak setup (see for instance Moreau (1990)). A tokamak is essentially a large transformer that drives a current through an axis symmetric plasma in a strong magnetic field (see e.g. Figure 5.1). The magnetic field in the toroidal direction is generated by the toroidal coils shown in Figure 5.1. Furthermore the transformer generates a magnetic field in the poloidal direction.

The total magnetic field in the toroidal direction, generated by the toroidal coils, in a tokamak is approximately \( 3T \). The current in the plasma, generated by the primary coil, can be of order \( 10^6 \text{A} \). This large electric current generates a magnetic field in the poloidal direction. This magnetic field is in general much smaller than the field in the toroidal direction, say 10% of the toroidal field. As a result the total
magnetic field vector $\mathbf{B}$ will describe helical patterns (see Figure 5.1) through the torus.

A magnetic field line, which is by definition closed, must reconnect after a number of passes through the torus. This reconnection can give rise to a geometric instability in the plasma. This instability will be strong when the number of passes needed for the reconnection is small and weak when the number of passes is large for the reconnection. Furthermore this instability will be dependent on the resistivity of the plasma. A high value of the resistivity, i.e. a low magnetic Reynolds number will have a stabilising effect on the instability.

In Figure 5.2 we show a sketch of a such a geometric instability in a resistive plasma. The patterns occurring in the poloidal plane are the so-called magnetic islands. These magnetic islands of instability are very similar to the so-called "Kelvin cateye" patterns which are found in normal fluids (see for instance Drazin & Reid (1981), Chandrasekhar (1961)).

The main goal of this chapter is to investigate whether the geometric instability which is observed in a plasma can also occur in normal fluids, described by the Navier-Stokes equations. For this we will use a geometry for the Navier-Stokes problem which is as close as possible to the geometry in the tokamak.

The geometry for the hydrodynamic problem consists of an outer torus which rotates in the toroidal and poloidal direction. Inside this torus an other (inner) torus is located which also rotates in both the toroidal and poloidal direction, however with different velocities as the outer torus. Due to this rotation and translation we will get a velocity field which describes helical pattern in the toroidal direction.
A streamline (particle trajectory) in such a velocity field will behave similar to a magnetic field line in a tokamak plasma. A sketch of the geometry is shown in Figure 5.3.

An important common element in both tokamak and fluid analogon is that $\nabla \cdot \mathbf{B}$ as well as $\nabla \cdot \mathbf{u}$. This means that field line equations as well as streamline equations can be written as Hamiltonians with two degrees of freedom: rotation in the toroidal direction and rotation in the poloidal direction. The Kolmogorov Arnold Moser (KAM) theorem, Arnold (1992), tells us that if the Hamiltonian for movement in both directions are decoupled one could expect that the streamlines lie on nested toroids as do the magnetic field lines in a stable tokamak plasma, i.e. the so-called magnetic surfaces. However, small coupling between the two directions in the Hamiltonians will according to the KAM-theorem lead to a break-up of the laminar behaviour and areas of three different characteristics will be created: islands (i.e. magnetic islands, cateye-patterns, etc.), unbroken toroids (KAM - surfaces) and stochastic areas around these islands in between the unbroken KAM-surfaces.

Such broken topology is recently proven to be the relevant one in the majority of tokamak plasmas (Lopez Cardozo et al. (1997)). The fascinating question would be if in a toroidal normal fluid analogon a similar KAM-pattern could be discovered. We will try to answer this question with help of numerical simulation.

The numerical model which we have used to solve the Navier-Stokes equation in the geometry given above, is basically the same as used for the calculation of curved pipe flow. Because of the full periodicity of the problem the constant pressure gradient in the axial direction, which we have used in chapter three and four has been removed. It is not necessary to resolve the region near the $r = 0$ axis an implicit time integration method is not needed. Therefore we have used the second order Adams-Bashforth method for the time integration which already has been described in chapter two.
Figure 5.3: The geometry and computational grid of the two toroids.

Figure 5.4: The coordinate system used, \( r \) denotes the radial coordinate, \( \theta \) the poloidal coordinate, and \( \phi \) the toroidal coordinate.
Table 5.1: The velocity at the wall of the inner (denoted by superscript i) and outer (denoted by superscript o) torus in the toroidal direction (denoted by subscript t) and the poloidal direction (denoted by subscript p), Re the Reynolds number defined as $U_i^o D/\nu$, and $N_i$ denotes the number of grid points in the ith direction.

<table>
<thead>
<tr>
<th>$U_i^o$</th>
<th>$0.6h_3/\kappa$</th>
<th>$0.6h_3/\kappa$</th>
<th>$6h_3/\kappa$</th>
</tr>
</thead>
<tbody>
<tr>
<td>$U_i^t$</td>
<td>$-0.6h_3/\kappa$</td>
<td>$-0.6h_3/\kappa$</td>
<td>$-6h_3/\kappa$</td>
</tr>
<tr>
<td>$U_p^o$</td>
<td>0.6</td>
<td>0.6</td>
<td>6</td>
</tr>
<tr>
<td>$U_p^t$</td>
<td>-0.6</td>
<td>-0.6</td>
<td>-6</td>
</tr>
<tr>
<td>$Re$</td>
<td>120</td>
<td>110</td>
<td>210</td>
</tr>
<tr>
<td>$N_r \times N_\theta \times N_s$</td>
<td>$33 \times 75 \times 175$</td>
<td>$33 \times 75 \times 175$</td>
<td>$33 \times 75 \times 175$</td>
</tr>
</tbody>
</table>

The results of the calculations have been visualised by means of a stream function and particle trajectories. The stream function for a given angle $\theta$ is defined as:

$$\psi(\theta) = \int_r^\rho w(r, \theta)dr.$$  \hspace{1cm} (5.3)

This function is used for the visualisation of the flow in the toroidal plane. In the poloidal plane contours of constant axial velocity are used for the visualisation. Furthermore particle trajectories have been calculated using the method proposed by Uijttewaal & Oliemans (1996).

5.3 Results of the simulations

In this section we will present some results obtained with the model described above.

We have chosen for an outer torus with an aspect ratio of $D/R = \kappa - 1/4$. The minor radius of the outer torus is equal to $D/2$ and of the inner torus equal to $D/8$. The velocity of the inner and outer torus in the toroidal and poloidal direction are given in Table 5.1. The velocities are chosen in such a way that

$$q = \frac{vR}{wr},$$  \hspace{1cm} (5.4)

is $O(1)$ in the area between the two toruses. In the sequel of this chapter it will be shown that this is an important parameter in the problem. Several calculations with different values of the Reynolds number have been performed (see e.g. Table 5.1). In all the calculations we used 33 points in the radial direction, 75 in the poloidal direction and 175 in the toroidal direction. The calculations were started...
from the initial conditions with zero velocity everywhere. At time zero the walls of both the inner and outer torus are set into motion. During the calculations integral quantities such as the bulk velocity and wall shear stress are monitored. After a certain time period these quantities become constant and the analysis of the results can be started.

To start with we show in Figure 5.5 plots of the velocity in the toroidal direction, normalised with the mean velocity in the toroidal direction. The curves are obtained at the poloidal position \(r = D/4, \theta = 0\) for values of the Reynolds number of 110 and 120 (see also Table 5.1). The high Reynolds number case \(Re = 120\) gives a harmonic fluctuating velocity as function of the toroidal angle \(\phi\), which is not the case for the low Reynolds number case. So there seems to be a sort of transition between a stable system and an unstable system for a Reynolds number somewhere in between 110 and 120. In the present investigation we are not interested in the exact value of the critical Reynolds number, but more in the flow patterns that occur in the torus. So we will not try to calculate the critical Reynolds number of the flow.

In Figure 5.6 a plot of the mean velocity in the poloidal and toroidal direction is shown (the profiles are averaged over the toroidal and poloidal direction) as a function of the minor radius. The velocity profile in the poloidal direction behaves more or less as a solid body rotation. The toroidal profile is over a large part of the radius also linear. It should be noted that the bulk velocity \(\overline{W}\) is not equal to zero and there will be a mean centrifugal force \(\rho\overline{W}^2/R\) acting on the fluid elements in the torus.

The time which a fluid element at a certain poloidal positions needs to make one complete turn through the torus in the toroidal direction is:

\[
t = \frac{2\pi R}{w},
\]

the time needed for \(q\) turns in the poloidal direction is:

\[
t = \frac{q2\pi R}{v}.
\]

Combining these relations gives the following relation for the quotient \(q\)

\[
q = \frac{vR}{wr}
\]

When \(q\) is zero there is thus no rotation in the poloidal direction and when \(q = \infty\) there is no rotation in the toroidal direction.

In Figure 5.7 a plot of \(q\) as function of the radius is shown. The value of \(q\) becomes very large for \(r/D \approx 0.2\), which means that the fluid at this position is only rotating
Figure 5.5: Top: The velocity in the toroidal direction at $r = D/2, \theta = \pi/2$. The solid curve has been calculated with a Reynolds number of 120 and the dashed curve with a Reynolds number of 110. Bottom: The power spectrum of the velocity ($Re = 120$).
Figure 5.6: The velocity profiles in the torus in the poloidal direction (solid line) and in the toroidal direction, averaged over the poloidal and toroidal directions.

Figure 5.7: The q profile as given by equation (5.4), averaged over the poloidal and toroidal directions.
Figure 5.8: A particle trajectory of fluid particles released at $r = 0.2D, 0.3D, 0.4D, 0.45D, \theta = 0$.

in the poloidal direction (e.g. Figure 5.6). The value of $q$ is zero for $r/D \approx 0.33$, i.e. fluid elements have only a velocity in the axial direction. This behaviour of the flow is qualitatively shown in Figure 5.8 where the particle trajectories of five particles are shown which are released at $\theta = 0$ and $r = 0.2, 0.3, 0.40, 0.45D$. The particle released at $r = 0.2D$ follows a path in the poloidal plane and likely reconnects with its own tail after some time. The other particles do not show such a possible reconnection.

The instability caused by the reconnection is also illustrated by the deviation of the stream function (equation (5.3)), which has been shown in Figure 5.9. The deviation of the stream function will be denoted by $\psi'$ in the sequel. The largest gradients in $\psi'$ are present close to the point where $r = 0.2$, i.e. the point where
Figure 5.9: The deviation of the mean value of the stream function, defined in section two (equation 5.3), plotted at $\theta = 0, \pi$. 
\[ q \to \pm \infty. \]

In Figures 5.10 and 5.11 we show contour plots of the deviation of the mean axial velocity in the poloidal plane for various positions at the torus. The toroidal distance between the subsequent plots is 0.25\(\pi\). The patterns shown in Figures 5.10 and 5.11 rotate clockwise. During one complete turn in the toroidal direction the patterns rotate almost a half turn in the poloidal direction. The number of cells shown in Figure 5.11 (four) is likely the same in every poloidal section. The number of cells in the toroidal direction shown in Figure 5.9 is equal to 14. So we find a ratio of \(14/4 = 3.5\).

In Table 5.1 also a simulation with a Reynolds number of 210 is listed. This simulation has a ten times higher rotation rate of the wall than the \(Re = 120\) simulation and also a higher value of the viscosity \(\nu\). Although the Reynolds number is of the same order the centrifugal force acting on the fluid elements will be much higher for this calculation, i.e. the Dean number (see chapter four) will be much higher.

In Figure 5.12 (top) a plot of the axial velocity at \(r = D/4, \theta = 0\) is shown. The figure shows that with increasing centrifugal force the wavelength of the instability decreases, compared to the results shown in Figure 5.5. In Figure 5.12 (bottom) the power spectrum of the velocity signal has been shown. If we compare this spectrum with the one given in Figure 5.5 it is clear that with increasing Reynolds number higher harmonics enter the problem. Increasing the Reynolds number further will lead to a turbulent flow.

In Figure 5.13 the \(q\)-profile for the \(Re = 210\) case has been shown. The value of \(q\) goes again to \(\pm \infty\) close to the inner torus.

In Figure 5.14 the deviation of the stream function and axial velocity for the \(Re = 210\) case has been shown. The deviations are the largest in the regions where \(q \to \pm \infty\). Furthermore, it is clear that with increasing centrifugal force also the number of cells increases (see also Figure 5.12). The ratio of cells in the toroidal direction to the poloidal direction is equal to \(24/7 \approx 3.5\). This ratio is the same as we have found for the case with the lower value of the centrifugal force.

### 5.4 Relation with a resistive plasma

In the previous section we have shown that there are instabilities present caused by geometric resonances above a certain value of the Reynolds number as there are in toroidal plasmas. However, the location of the "cataeyes" corresponds with a \(q\) value which is infinite. Moreover the extension of the instabilities is over the full cross-section Some streamline are closed without one or more passes along the torus in the toroidal direction. In a resistive plasma also instabilities are observed
Figure 5.10: The deviation of the mean axial velocity plotted in the poloidal plane. The plots are obtained at the toroidal angle 0.25, 0.5, 0.75, 1(\pi).
Figure 5.11: The deviation of the mean axial velocity plotted in the poloidal plane. The plots are obtained at the toroidal angle $1.25, 1.5, 1.75, 2\pi$. 
Figure 5.12: Top: The axial velocity at $r = D/4, \theta = 0$ along the torus. Bottom: Power spectrum of the axial velocity at $r = D/4, \theta = 0$ along the torus.
which closes on themselves after a certain number of passes in the poloidal and toroidal direction. The ratio of these corresponds to the local $q$-value as predicted by the KAM-theorem. Moreover the island width is small, which is not the case for the Navier-Stokes problem. The instabilities in such a plasma are therefore of a different form than the instability we observe.

The reason for the failure of the analogy, is probably due to the presence of the centrifugal force

$$F_c = \frac{\rho W^2}{R}$$

which acts on all the elements of the flow. This force will generate a strong double vortex in the poloidal plane, which will be similar to the Dean vortices shown in chapter four. The coupling between the poloidal and toroidal direction in the streamline Hamiltonian is therefore strong instead of weak as supposed by the KAM-theorem.

This centrifugal force and thus the secondary motion is of course absent in the equation for the magnetic induction $B$. The influence of the secondary motion (or the centrifugal force), will always be stronger than the weak instability. Therefore, it is probably not possible to simulate numerically, the same topology in normal fluids as in resistive plasma experiments not withstanding the many mathematical similarities between the two systems.

As last point we may mention that the cells we observe in the torus are related to the so-called Bernard-cells, Drazin & Reid (1981), which are observed in thermal
driven fluids. In such a fluid the buoyancy forces acts on the fluid elements and cell structures similar to the one we observed in the torus are found. So both Bernard-cells and the cells found in the torus are driven by a volume force, i.e. the buoyancy force and the centrifugal force respectively. Therefore we feel that the cells we observe in the torus are more related to the Bernard cells than to the cells observed in plasma experiments.

5.5 Conclusion

In this chapter we have studied the apparent analogy between the flow of an ordinary fluid and the magnetic induction in a resistive plasma. The analogy has been used to calculate the flow of a resistive plasma in a tokamak setup. In both the plasma and the fluid systems a geometric resonance occurs. However the length scale of the observed instabilities is different, which is caused by the centrifugal force acting on the flow which is not present in the equations governing the plasma flow. It is not possible to eliminate the centrifugal force from the fluid problem by a proper choice of the boundary conditions for the velocity. (Removing the Christoffel terms in the equations presented in chapter three would give better results, but this if from a physical point of view not acceptable.)

We conclude that it is not possible to predict the instabilities in a plasma using the mathematical analogy which exists between the viscous fluid flow and the flow of a plasma. Therefore we also expect that the turbulence in a plasma has a different nature then the turbulence in an ordinary fluid.
Figure 5.14: Plots of the deviation of the stream function $\psi'$ and the deviation of the axial velocity for a Reynolds number of 210.
Chapter 6

Discussion and Conclusion

In the last chapter of this thesis we will summarise the main results and conclusions based on the work presented in this thesis. Occasionally, a few recommendations for further research will be given.

The first aim of the work presented in this thesis has been to accurately describe the flow in an electromagnetic flow meter and to determine the electric potential distribution occurring in such a device. Although electromagnetic flow meters have been used for more than four decades in practical applications, to our knowledge no accurate experimental data are available, which relate the turbulent flow in this device to the measured (electric) signal. To obtain such information we have carried out numerical simulations in combination with experiments.

For the experimental part we have modified an existing pipe setup in such a way that is possible to measure the electric potential at wall induced by a strong homogeneous magnetic field. The main goal of the experiment was to obtain data to validate the numerical simulation. Therefore we restricted ourselves to a simple case, i.e. fully developed straight pipe flow in combination with a homogeneous magnetic field. In the experiments only integral quantities of the flow have been measured, such as the pressure gradient and the bulk velocity. The electric potential as a function of time has been measured at eight different locations on the pipe circumference. These latter data are used for comparison with the numerical results.

For the numerical simulation of the flow meter we have used exactly the same geometry as in the experiment, in particular with respect to magnetic field. The flow is computed with help of Large Eddy Simulation (LES). The values of the input parameters for the LES are obtained from the experiment. In the LES the electric potential at the wall has been recorded in a similar way as in the experiment. The first and second-order moments of both the simulated and measured electric
potential are in very good agreement. LES seems therefore a very useful tool to predict the response of a flow meter in turbulent flow. Furthermore, the spectra of the electric potential obtained from LES and experiments, show that the electric potential is mainly generated by the large eddies in the flow which have typically a length scale of 5% of the pipe diameter or more. This explains the excellent performance of LES. Namely, LES is known to resolve the large scales in the flow accurately while the modelling of the small scales is still a problem in LES, especially near walls. This shortcoming of LES will be of no consequence in this particular case.

Although, only results for homogeneous magnetic fields are presented in this thesis it should be stressed that the extension of the modelling to non-homogeneous magnetic fields is trivial. Based on the results presented here, we are fully confident that the same accuracy of the simulations can be expected.

The second aim of our research has been to investigate the influence of a disturbance in the flow on the reading of a flow meter. Again we have restricted ourself to a simple case, namely the flow in a pipe bend. Such a disturbance is reasonable simple to model and is of practical importance as most piping systems contain bends. To calculate the flow in a pipe bend we have reformulated the governing equations in a toroidal coordinate system. With the model formulated in this system we have first investigated the flow patterns occurring in a bend by means of Direct Numerical Simulation. The results of these simulations show that for certain ranges of the Dean and Reynolds numbers multiple solutions of the governing equations exists. This observation is supported by flow visualisations studies in a related geometry, i.e. the flow in a curved duct.

With help of the numerical simulations, the response of a flow meter for the flow in the bend has been calculated. It is shown that the influence of the curvature on the reading of the flow meter is smaller than the effect of the bend on the flow patterns and turbulence. For the configuration studied in this thesis we have found an error of approximately 5% for the case when the flow meter was placed in the bend. For the practical application of flow meters this is certainly a non-negligible error. However, in practice flow meters are always located downstream of a bend so the error will be smaller. Furthermore, it has been shown that the error in the reading of the flow meter can be reduced by changing the orientation of the magnetic field.

Finally, in chapter five the mathematical analogy between resistive plasmas and viscous fluids has been studied. From the equations presented in chapter two for the magnetic induction $\mathbf{B}$ and the vorticity, i.e. the rotating of the velocity vector, it is clear that there is a mathematical analogy. We have used this analogy to study a the transition of the plasma, in a tokamak setup, from a laminar to a turbulent. It is of interest if this transition in a plasma has the same origin as the transition
in an ordinary fluid. If this would be the case then the existing theory for fluid turbulence could be used to describe plasma turbulence.

To investigate this we performed calculations for normal fluids in a geometry which is very similar to the geometry found in plasma experiments. It has been shown that in the fluid system geometric resonances are present and so-called cat-eye patterns occur. These resonances and cat-eye patterns are also observed in plasma experiments. However, the length scale of the cat-eyes is very different for both systems. In the plasma case small cat-eyes are present which have a typical length scale of 5% of the minor radius of the torus, while in the viscous case these patterns fill the whole cross section of the torus. It has been argued that this difference is caused by the centrifugal force. This force is acting in the fluid analogon, while it is absent in the resistive plasma. Therefore the analogy between the flow of a plasma and the flow of an ordinary fluid is not complete.
A Analytic Solution of the flow meter equation

In this section we will investigate the effect of turbulence on the electric potential, resulting from equation (3.1). The two dimensional Poisson equation reads:

\[
\frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial V}{\partial r} \right) + \frac{1}{r^2} \frac{\partial^2 V}{\partial \theta^2} = B \sin \theta \frac{1}{r} \frac{\partial w}{\partial \theta} + B \cos \theta \frac{\partial w}{\partial r}. \tag{A.1}
\]

In the following we will assume that the electric potential \(V\) and the axial velocity \(w\) can be written as

\[
V = \sum_n v_n \cos n\theta, \quad w = \sum_m w_m \cos m\theta \tag{A.2}
\]

Substituting these relations in equation (A.1) gives:

\[
\sum_n [(v_n'' + \frac{1}{r} v_n' - \frac{n^2}{r^2} v_n) \cos n\theta = \sum_m \left[ -\frac{m}{r} w_m B \sin \theta \sin m\theta + B \cos \theta \cos m\theta \frac{\partial w_m}{\partial r} \right] \tag{A.3}
\]

where a prime denotes differentiation with respect to \(r\). The solution of the homogeneous differential equation is given by:

\[
v_n(r) = c_n r^n + d_n r^{-n} \tag{A.4}
\]

where \(c_n\) and \(d_n\) are integration constants. The terms at the right hand side of equation (A.3) can be simplified using the following relations

\[
2 \cos \theta \cos m\theta = \cos(\theta - m\theta) + \cos(\theta + m\theta)
\]

\[
2 \sin \theta \sin m\theta = \cos(\theta - m\theta) - \cos(\theta + m\theta). \tag{A.5}
\]

Furthermore for simplicity we will assume that \(B\) is equal to unity. For the particular solution of equation (A.3) we need the Wronskian of the homogeneous differential equation (A.4). The Wronskian of \((r^n, r^{-n})\) is equal to \(-2n/r\). With help of the orthogonality of the trigonometric functions and the Wronskian, we find the following relations between the velocity components \(w_m\) and the electric potential \(v_n\)

\[
1\text{The particular solution of a second order differential equation } y''(r) + ay'(r) + by(r) = f(r) \text{ with } \phi_1(r) \text{ and } \phi_2(r) \text{ as solutions of the homogeneous differential equation is given by}
\]

\[
y_p = \int_{r}^{r} \left( \frac{\phi_1(t)\phi_2(r) - \phi_1(r)\phi_2(t)}{W[\phi_1(t), \phi_2(t)]} \right) f(t) dt
\]
\[ v_1(r) = c_1 r + d_1 r^{-1} + \frac{1}{4} \left( r \int_{t}^{R} t^{-1} \left[ t \frac{\partial w_2}{\partial t} + 2t \frac{\partial w_0}{\partial t} - 2w_2 \right] dt - r^{-1} \int_{0}^{r} t \left[ t \frac{\partial w_2}{\partial t} + 2t \frac{\partial w_0}{\partial t} - 2w_2 \right] dt \right) \] (A.6)

\[ v_n(r) = c_n r^n + d_n r^{-n} + \frac{1}{4n} \left( r^n \int_{\infty}^{t} t^{-n} \left[ t \frac{\partial w_{n+1}}{\partial t} + (n-1)w_{n-1} - (n+1)w_{n+1} + t \frac{\partial w_{n-1}}{\partial t} \right] dt - r^{-n} \int_{0}^{r} t^n \left[ t \frac{\partial w_{n+1}}{\partial t} + (n-1)w_{n-1} - (n+1)w_{n+1} + t \frac{\partial w_{n-1}}{\partial t} \right] dt \right) \] (A.7)

It will be clear that \( d_n \) should be zero for a bounded solution at the origin. Let us assume that the flow is axis-symmetric and that the pipe wall is isolated in this case

\[ w_0 \neq 0, \quad \text{and}, \quad w_{1,2,3,..} = 0. \] (A.8)

Thus

\[ v_1(r) = c_1 r + \frac{1}{4} \int_{0}^{r} 2 \frac{\partial w_0}{\partial t} dt - \frac{1}{4r} \int_{0}^{r} 2t^2 \frac{\partial w_0}{\partial t} dt \] (A.9)

Furthermore \( \partial v_1 / \partial r \) should be zero at the pipe wall \( (r = R) \), thus

\[ 0 = v_1'(R) = c_1 + \frac{1}{4} \int_{0}^{R} 2 \frac{\partial w_0}{\partial t} dt + \frac{1}{4} R \left( 2 \frac{\partial w_0(R)}{\partial r} \right) - \frac{1}{4R} \left( 2R^2 \frac{\partial w_0(R)}{\partial r} \right) \] (A.11)

thus

\[ c_1 = \frac{1}{2} w_0(0) + \frac{1}{2R} \int_{0}^{R} w_0 dt \]

and \( v_1(r) \) reads:

\[ v_1(r) = \frac{1}{2} rw_0(0) + \frac{r}{R^2} \int_{0}^{r} w_0 dt + \frac{1}{4} \int_{0}^{r} 2 \frac{\partial w_0}{\partial t} dt - \frac{1}{4r} \int_{0}^{r} 2t^2 \frac{\partial w_0}{\partial t} dt \] (A.12)

When the equation above is integrated from the centerline of the pipe to the wall, most terms at the right hand side vanish. The thirth term at the right hand side
of equation (A.12) is equal to zero and the second term cancels the fourth (after partial integration). So \( v_1(R) \) becomes \( R w_0(0)/2 \).

The electric potential at the pipe wall is again given by:

\[
V(R, \theta) = BRU_b \cos \theta. \tag{A.13}
\]

In which

\[
U_b = \frac{1}{\pi R^2} \int_0^{2\pi} \int_0^R wr dr d\theta = \frac{2}{R^2} \int_0^R wr dr
\]

Which is exactly the same as found in chapter 3, equation (3.4).

Let us now assume that the velocity field is turbulent. In this case \( w_1, w_2, \ldots \) are not zero anymore. If we exclude the part of the solution \( r > R \), equation (A.7) reads

\[
v_n(r) = c_n r^n + r^{-n} \int_0^r t^n [t \frac{\partial w_{n+1}}{\partial t} + (n-1)w_{n-1} - (n+1)w_{n+1} + t \frac{\partial w_{n-1}}{\partial t}] dt \tag{A.14}
\]

After partial integration we obtain

\[
v_n(r) = c_n r^n + \frac{1}{4n} \left( r w_{n+1} + \frac{n-1}{n+1} r w_{n-1} - r w_{n+1} - r w_{n-1} + O(\frac{1}{n}) \right) \tag{A.15}
\]

or

\[
v_n(r) = c_n r^n + O(n^{-2}). \tag{A.16}
\]

In the inertial subrange of the flow it is known then the energy spectrum, thus the spectrum of \( w \) behaves like some power of \( n \), say

\[
\nu^2 \propto \nu \epsilon^{2/3} n^{-5/3},
\]

Which is the well known \( k^{-5/3} \) law Tennekes & Lumley (1974). When we substitute this relation in equation (A.7) and neglecting small terms we can write

\[
v_n(r) \approx \frac{1}{n^2} r^n r^{-n} r \epsilon^{1/3} n^{\sqrt{-5/3}}
\]

thus

\[
\nu^2 \propto \frac{1}{n^4} \epsilon^{2/3} n^{2-5/3}
\]

and the spectrum of \( v_n \) should have a slope of \( n^{-17/3} \). This slope is different as observed in the experiments reported in chapter three.
A.1 Some remarks on the 3D case

In the previous section we have seen that a two dimensional solution does not give a correct behaviour for the spectrum of $\tilde{v}$. This is probably caused by the 3D character of turbulence. Let us therefore consider the flow meter equation in three dimensions. We will still assume that the magnetic field is two dimensional and that the velocities in the radial and circumferential direction are small compared to the axial velocity. So we write for $V, w$

$$V = \sum \sum v_{n,k} \cos \theta \exp(ikz), \quad w = \sum \sum w_{n,k} \cos \theta \exp(ikz).$$

Here $k$ is the wave number in the axial direction, and we have assumed that the potential is periodic in the axial direction. For this case the solutions of the homogeneous differential equation for $v_{n,k}$ are $I_n(kr)$ and $K_n(kr)$, where $I_n$ and $K_n$ are modified Bessel functions (e.g. Abramowitz & Stegun 1972). These functions have qualitative the same behaviour as the solutions $r^n$ and $r^{-n}$. The Wronskian of $K_n(kr)$ and $I_n(kr)$ is equal to $1/r$ and the solution of Poisson’s equation in three dimensions reads

$$v_n(r) = c_n I_n(kr) + d_n K_n(kr) + \frac{1}{2} I_n(kr) \int_0^r \frac{K_n(t)}{t} f(t) dt - \frac{1}{2} K_n(kr) \int_0^r \frac{I_n(t)}{t} f(t). \tag{A.17}$$

where $f(t)$ is the right hand side of the Poisson equation. The solution at $r = 0$ has to be bounded thus $d_n$ must be zero. Partial integration gives of the term valid for $r < R$ gives

$$\int_0^r t^{-1} I_n(t) f(t) dt = f(R)[I_{n-1}(r) - (2\Gamma(2))^{-1}] + H.O.T.$$ 

So the integral in equation (A.17) behaves as

$$K_n(kr) I_{n-1}(r)$$

For large values of $n$ these term can be approximated by (Abramowitz & Stegun (1972)):

$$\frac{1}{\sqrt{2\pi n}} \sqrt{\frac{\pi}{2(n-1)}} \approx \sqrt{\frac{1}{4n(n-1)}} \approx \frac{1}{2n}$$

If we again assume that the flow has a $n^{-5/3}$ spectrum we find for

$$\tilde{v}^2 \propto \left( \frac{1}{2n} \right)^2 \epsilon^{2/3} n^{-5/3}$$

and the spectrum of $v_n$ should have a slope of $n^{-11/3}$. This slope agrees reasonable with the experiment reported in chapter three.
Solution of Poissons’s equation

In this section we will point out how the three dimensional Poisson equation can be solved numerically. The three dimensional Poisson operator reads:

\[
\nabla \cdot (\nabla f) = \frac{1}{h_1 h_2 h_3} \cdot \left[ \frac{\partial}{\partial \eta_1} \left( \frac{h_2 h_3}{h_1} \frac{\partial f}{\partial \eta_1} \right) + \frac{\partial}{\partial \eta_2} \left( \frac{h_1 h_3}{h_2} \frac{\partial f}{\partial \eta_2} \right) + \frac{\partial}{\partial \eta_3} \left( \frac{h_1 h_2}{h_3} \frac{\partial f}{\partial \eta_3} \right) \right] = g(\eta_1, \eta_2, \eta_3)
\]

(B.1)

Let us suppose that

\[
\frac{h_2 h_3}{h_1} \neq f(\eta_1)
\]

Then equation (B.1) can be rewritten as

\[
\frac{h_1 h_2 h_3 g(\eta_1, \eta_2, \eta_3)}{h_1} = \frac{h_2 h_3}{h_1} \frac{\partial^2 f}{\partial \eta_1^2} + \frac{\partial}{\partial \eta_2} \left( \frac{h_1 h_3}{h_2} \frac{\partial f}{\partial \eta_2} \right) + \frac{\partial}{\partial \eta_3} \left( \frac{h_1 h_2}{h_3} \frac{\partial f}{\partial \eta_3} \right)
\]

(B.2)

The term \(\frac{\partial^2 f}{\partial \eta_1^2}\) can be discretised with help of second order central differences, e.g.

\[
\frac{\partial^2 f}{\partial \eta_1^2} = \frac{f_{i+1} - 2f_i + f_{i-1}}{\Delta \eta_1^2}
\]

When the boundary conditions in the \(\eta_1\) direction are periodic, we can use the following Fourier Transforms to reduce the difference equation to and ordinary equation (Fletcher 1990)

\[
h_{i,j,k} = \sum_{s=1}^{N} H_{s,j,k} \exp \left[ \frac{s \sqrt{-1} \pi}{N} \right],
\]

Which yields the following equation

\[
\sum_{s=1}^{N} h_1^2 g(s, \eta_2, \eta_3) \exp \left[ \frac{s \sqrt{-1} \pi}{N} \right] = \lambda_s f + \frac{1}{h_1 h_2 h_3} \left[ \frac{\partial}{\partial \eta_2} \left( \frac{h_1 h_3}{h_2} \frac{\partial f}{\partial \eta_2} \right) + \frac{\partial}{\partial \eta_3} \left( \frac{h_1 h_2}{h_3} \frac{\partial f}{\partial \eta_3} \right) \right]
\]

(B.3)

for \(s = 1, ..., N\).

In which \(\lambda\) is the eigenvalue of the Fourier Transform. For a staggered grid with periodic boundary conditions this eigenvalue is equal to

\[
\lambda_s = -\frac{4}{\Delta \eta_1^2} \sin^2 \left[ \frac{(s-1) \pi}{2N} \right].
\]
With the method outlined above the three dimensional Poisson equation is reduced to $N$ two dimensional Helmholtz equations. These two dimensional equations can be solved using an iterative technique like (CG Golub & van Loan (1993)), or with a direct method like cyclic reduction, Golub & van Loan (1993). Sometimes, depending on the co-ordinate system used, a reduction to a one dimensional problem is possible. Discretization of the one dimensional problem leads to a tridiagonal matrix which can be solved very efficiently with a Thomas algorithm (Golub & van Loan (1993)). The solutions of the one or two dimensional problems have to be transformed with the inverse of the Fourier Transform which yields the solution of the three dimensional problem.

For non-periodic boundary conditions different transformations have to be used, see for instance Swarztrauber (1977) or Sweet (1973). In principle all combinations of Neumann and Dirichlet boundary conditions are possible.
REFERENCES

Abramowitz, M. and Stegun, I.A. (eds.) (1972), Handbook of mathematical functions, Dover, New York


Brigham, O., The Fast Fourier Transform, Pretence Hall.


Chapter 6


Milligen, B.P. van, 1991, Analysis of equilibrium and topology of tokamak plasmas,
Elinkwijk, Utrecht, 1991


Orlandi, P. & Fatica, M., 1995, Direct simulation of a turbulent pipe rotating along the
axis, Submitted to : J. Fluid Mech..


Press.
257-271.

Piomelli, U., Ferziger, J. and Moin, P., 1988, Model consistency in large eddy simulation
of turbulent channel flows, Physics of Fluids, 31, pp. 1884-1891

Pourquié, M., 1994, Large-eddy Simulation of a Turbulent Jet, PhD-thesis Delft University
of Technology, The Netherlands.

Ramshankar, R. & Sreenivasan, K.R., 1988, A paradox concerning the extended Stokes

Rowe, M., 1970, Measurement and computations of flow in pipe bends, J. Fluid Mech. 43,
771-783

Shercliff, J.A., 1962, The theory of electromagnetic flow-measurement Cambridge University

Schumann, 1973,Ein Verfahren zur direkten numerische simulation turbulenten strömungen
in platten- und ringspaltkanälen und über seine anwendung zur untersuchung von
turbulenzmodellen, PhD-thesis, University of Karlsruhe, Germany.


Swarztrauber, P., 1977, The methods of cyclic reduction, Fourier analysis and the FACR
algorithm for the discrete solution of Poisson’s equation on a rectangle, SIAM Review,
19, 490-501.

R. Sweet, Direct methods for the solution of Poisson's equation on a staggered grid, J. of

A., 124, 243-249. (also in: The collected paper of G.I. Taylor).

MA, USA.

Toonder J. den., 1996, Drag Reduction by Polymer Additives in a Turbulent pipe flow:
Laboratory and Numerical Experiments, PhD Thesis Delft University of Technology, Delft, The Netherlands
Curriculum Vitae


Na zijn afstuderen begon hij in augustus 1993 aan zijn promotie op het Laboratorium voor Aero en Hydrodynamica van de TU Delft. De titel van het promotie onderzoek was Electro-magnetic effects in turbulent pipe flows: a numerical simulation with application to an electro-magnetic flowmeter en werd uitgevoerd onder de supervisie van Prof.dr.ir. F.T.M. Nieuwstadt.
